PC-DMIS Laser Manual

For Version 2017 R1

Generated November 22, 2016
Hexagon Manufacturing Intelligence

lpsolve information

PC-DMIS uses a free, open source package called lp_solve (or lpsolve) that is distributed under the GNU Lesser General Public License (LGPL).

```
lpsolve citation data
----------------------
Description: Open source (Mixed-Integer) Linear Programming
system
Language: Multi-platform, pure ANSI C / POSIX source code,
Lex/Yacc based parsing
Official name: lp_solve (alternatively lpsolve)
Release data: Version 5.1.0.0 dated 1 May 2004
Co-developers: Michel Berkelaar, Kjell Eikland, Peter Notebaert
Licence terms: GNU LGPL (Lesser General Public Licence)
Citation policy: General references as per LGPL
Module specific references as specified therein
You can get this package from:
http://groups.yahoo.com/group/lp_solve/
```

Crash Reporting Tool

PC-DMIS uses this crash reporting tool:

"CrashRpt"

MERCHANTABILITY AND FITNESS FOR A PARTICULAR PURPOSE ARE DISCLAIMED. IN NO EVENT SHALL THE COPYRIGHT HOLDER OR CONTRIBUTORS BE LIABLE FOR ANY DIRECT, INDIRECT, INCIDENTAL, SPECIAL, EXEMPLARY, OR CONSEQUENTIAL DAMAGES (INCLUDING, BUT NOT LIMITED TO, PROCUREMENT OF SUBSTITUTE GOODS OR SERVICES; LOSS OF USE, DATA, OR PROFITS; OR BUSINESS INTERRUPTION) HOWEVER CAUSED AND ON ANY THEORY OF LIABILITY, WHETHER IN CONTRACT, STRICT LIABILITY, OR TORT (INCLUDING NEGLIGENCE OR OTHERWISE) ARISING IN ANY WAY OUT OF THE USE OF THIS SOFTWARE, EVEN IF ADVISED OF THE POSSIBILITY OF SUCH DAMAGE.

nanoflann Library

PC-DMIS uses the nanoflann library (version 1.1.8). The nanoflann library is distributed under the BSD License:

Software License Agreement (BSD License)

Copyright 2008-2009 Marius Muja (mariusm@cs.ubc.ca). All rights reserved.

Copyright 2008-2009 David G. Lowe (lowe@cs.ubc.ca). All rights reserved.

Copyright 2011 Jose L. Blanco (joseluisblancoc@gmail.com). All rights reserved.

THE BSD LICENSE

Redistribution and use in source and binary forms, with or without modification, are permitted provided that the following conditions are met:

1. Redistributions of source code must retain the above copyright notice, this list of conditions and the following disclaimer.

2. Redistributions in binary form must reproduce the above copyright notice, this list of conditions and the following disclaimer in the documentation and/or other materials provided with the distribution.

THIS SOFTWARE IS PROVIDED BY THE AUTHOR "AS IS" AND ANY EXPRESS OR IMPLIED WARRANTIES, INCLUDING, BUT NOT LIMITED TO, THE IMPLIED WARRANTIES OF MERCHANTABILITY AND FITNESS FOR A PARTICULAR PURPOSE ARE DISCLAIMED. IN NO EVENT SHALL THE AUTHOR BE LIABLE FOR ANY DIRECT, INDIRECT, INCIDENTAL, SPECIAL, EXEMPLARY, OR CONSEQUENTIAL DAMAGES (INCLUDING, BUT NOT LIMITED TO, PROCUREMENT OF SUBSTITUTE GOODS OR SERVICES; LOSS OF USE, DATA, OR PROFITS; OR BUSINESS INTERRUPTION) HOWEVER CAUSED AND ON ANY THEORY OF LIABILITY, WHETHER IN CONTRACT, STRICT LIABILITY, OR TORT (INCLUDING NEGLIGENCE OR OTHERWISE) ARISING IN ANY WAY OUT OF THE

NLopt Library

PC-DMIS uses the NLopt library (2.4.2). The NLopt library is distributed under the GNU Lesser General Public Licence.

NLopt has this main copyright:

NLopt also contains additional subdirectories with their own copyrights that are too numerous to list here (see the subdirectories on this project page: https://github.com/stevengj/nlopt).

Qhull Library

PC-DMIS uses the Qhull library (2012.1):

and

The National Science and Technology Research Center for Computation and Visualization of Geometric Structures

(The Geometry Center)

University of Minnesota

email: qhull@qhull.org

This software includes Qhull from C.B. Barber and The Geometry Center.

Qhull is copyrighted as noted above. Qhull is free software and may be obtained via http from www.qhull.org. It may be freely copied, modified, and redistributed under the following conditions:

1. All copyright notices must remain intact in all files.

2. A copy of this text file must be distributed along with any copies of Qhull that you redistribute; this includes copies that you have modified, or copies of programs or other software products that include Qhull.

3. If you modify Qhull, you must include a notice giving the name of the person performing the modification, the date of modification, and the reason for such modification.

4. When distributing modified versions of Qhull, or other software products that include Qhull, you must provide notice that the original source code may be obtained as noted above.

5. There is no warranty or other guarantee of fitness for Qhull, it is provided solely "as is". Bug reports or fixes may be sent to qhull_bug@qhull.org; the authors may or may not act on them as they desire.

Eigen Library

PC-DMIS uses the Eigen Library. This library is primarily licensed under the Mozilla Public Library Version 2.0 (MPL2) license (https://www.mozilla.org/en-US/MPL/2.0/) and partly licensed under the GNU Lesser General Public Licence (LGPL). For more information, see Licensing at http://eigen.tuxfamily.org.

RapidJSON Information

PC-DMIS uses the RapidJSON software package. The software is used and distributed under this MIT license:

Protocol Buffers Information

PC-DMIS uses Google's protocol buffers mechanism. The code is used and distributed under the terms of this license:

Non-Negative Least Squares

PC-DMIS uses the Non-Negative Least Squares Algorithm for Eigen:

ZeroMQ libzmq 4.0.4 Library

Freeicons.png Information

These icons from freeiconspng.com are used in our help documentation:

- eye icon
- computer icon

lightbulb icon

Table Of Contents

Using PC-DMIS Laser

PC-DMIS Laser: Introduction

This documentation covers how to use PC-DMIS with your laser sensor to measure features on a part or to collect data. Laser sensors allow you to collect millions of points of data in one or more cloud of points (COP). These clouds of points are then used within PC-DMIS for surface contour maps, export to reverse engineering packages, and creation of constructed features and auto features. This documentation discusses how to use PC-DMIS with a non-contact laser sensor to collect and interpret those clouds of points.

PC-DMIS Laser supports these hardware configurations:

- Perceptron - Digital, V4, V4i, V4ix, and V5
- HP-L-10.6 (CMS106) for DCC and HP-L-20.8 for DCC and Portable

HINT: You can use the CMS108 on both DCC and Portable machines.

The main topics in this help document include:

- Attributes for Laser Measurement
- Getting Started
- Using the Probe Toolbox in PC-DMIS Laser
- Execution Modes
- Using Sound Events
- Using the Laser View
- Using the Scan Line Indicator
- Understanding the Visualization Tools
- Pointcloud Scanning Colors
- Using the Laser Toolbars
- Using Pointclouds
- Creating a Mesh Feature
- Caliper Overview
- Pointcloud Operators

- Pointcloud Alignments
- TCP/IP Pointcloud Server
- Extracting Auto Features from Pointclouds
- Creating Auto Features with a Laser Sensor
- Clearing Auto Feature Scan Data
- Scanning Your Part Using a Laser Sensor
- Handling Laser Sensor Errors with ONERROR

If you come across something in the software that isn't covered in this documentation, see the main PC-DMIS documentation.

Attributes for Laser Measurement

Before getting into the details of non-contact laser sensors, you need to understand their attributes in order to improve the results obtained when using them for measurement. Laser sensors are excellent at gathering large amounts of data quickly. They are also good for measuring parts that may otherwise deform under the pressure of a tactile probe.

However, keep in mind that other factors, such as sunlight, surface finish, surface reflectivity, and surface color can influence measurements taken with laser sensors. In order to compensate for some of these factors, you can apply filters to the data to handle the influence. But you should understand how and why these items affect the measurement results.

Unlike other non-contact systems, laser sensors are not generally affected by standard industrial lighting. Laser sensors work under varied lighting conditions because the sensor’s frequency is tuned to its own laser. Only light that has the same frequency as the laser itself can affect the measurement. Because sunlight contains all frequencies of light, it is important to keep sunlight out of the inspection room.

Because tactile probes are larger than the deviation in most surface finish, a tactile probe acts as an averaging filter. When the tactile probe comes in contact with the surface, it gives an average of the highest points on the surface. When using a laser sensor, the light reflects off the surface of the part. How the light reflects depends a lot on the roughness of the surface, even if it does not appear rough to human sight or touch.

Generally, surfaces with a matte finish work better than those with a glossy finish. A glossy surface finish usually has directional reflection. Based on the angle of the light, you can get too much or too little light. You might even get a ‘hot spot’ (something that looks like a ‘blob’ in the graphics display area). This *blob* is actually the image of the light source. The reflection of light might add some extraneous points to the scan line, but the rest of the points are not affected by the reflection. To compensate for surface reflectivity, you can spray the part with an aerosol powder or paint.

Because the laser is light, the surface color can potentially impact the measurement. Similar to how something colored black absorbs heat from the sun, black surfaces on a part absorb the laser's light and make the measurement of those surfaces difficult. Darker colors have more potential for problems than lighter colors. If your part is too dark, you can apply powder coatings to it to make it easier to sample.

It usually takes time and experience working with your particular parts and in your specific environment to determine what settings work best for you. You should experiment with the capabilities of your specific sensor to improve measurement results.

WARNING: Exercise caution when working with Laser sensors as they can damage your eyes. Consult your laser sensor documentation for safety issues and procedures for a safe work environment.

Getting Started

Before you use PC-DMIS with your laser device, the basic steps below can help you verify that your system is properly prepared.

To get PC-DMIS running with your laser sensor, follow these steps:

If you are using a Perceptron laser on a Romer arm, see the "Using a Romer Portable CMM" section in the PC-DMIS Portable documentation.

Step 1: Install and Launch PC-DMIS

Before you use your laser device, ensure that PC-DMIS has been properly installed on your computer system.

To install PC-DMIS for your laser device:

1. Ensure that the machine that runs the laser sensor is properly setup and configured according to your machine’s specifications. Follow the documentation that came with your laser sensor to properly connect the hardware.
2. Ensure that you have a license (or portlock) that supports the Laser option. This tells the installer to install the necessary Laser components. If you don't have the necessary license (or a properly configured portlock), please contact your PC-DMIS software distributor.
3. Follow the instructions in the readme.pdf file and install PC-DMIS.

4. Start PC-DMIS in online mode by selecting **Start | All Programs | <Version> | <Version> Online**, where <version> represents your version of PC-DMIS.
5. Open an existing measurement routine, or create a new one. If you create a new measurement routine, the **Probe Utilities** dialog box appears so you can define your laser sensor in the next step.

NOTE: The PC-DMIS installer manages the installation of drivers, and so on.

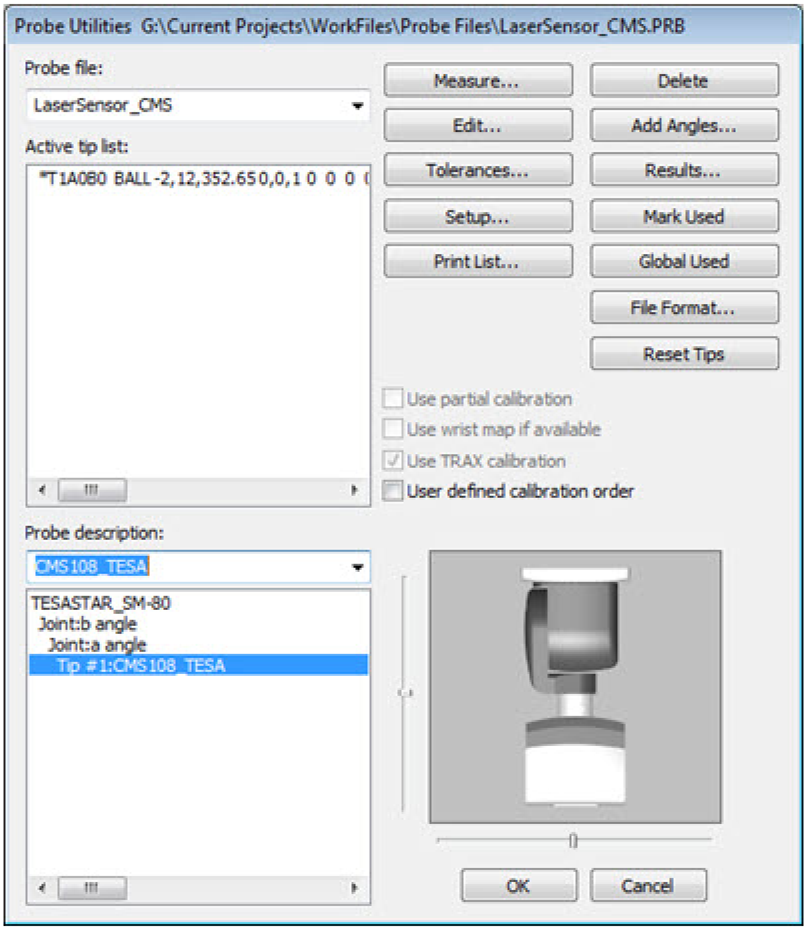
Setting Parameters without a Measurement Routine

Some users may need the ability to change laser parameters without first opening a measurement routine. If needed, you can access the **Laser Sensor** tab for the current laser sensor in the **Setup Options** dialog box by pressing the F5 key or selecting **Edit | Preferences | Setup**. The **Laser Sensor** tab is discussed in Step 3.

Step 2: Define the Laser Sensor

If you don't have a defined laser sensor, use the **Probe Utilities** dialog box to define it. This creates a probe file.

1. Select the **Insert | Hardware Definition | Probe** menu option to open the **Probe Utilities** dialog box. (This dialog automatically appears whenever you create a new measurement routine.)



Probe Utilities dialog box

2. In the **Probe file** box, type a name that best describes your laser sensor.
3. From the list of components at the bottom, select the **No probe defined** text to highlight it.
4. From the **Probe Description** list, select the appropriate probe. Most laser sensors connect directly to the *PH10M* probe head. A CMS108 sensor that you use with a DCC machine connects to a Tesastar probe head.
5. As needed, select additional components in the same manner for "empty connections" until you finish defining the probe. A defined probe shows a tip in the **Active Tip List**.

> **HINT:** Once you define the tip, the software no longer shows the probe image. This makes it so that the graphical image of the probe does not obstruct the view of the part during measurement. However, if you want to enable the display of probe components, double-clicking on the probe component to open the **Edit Probe Component** dialog box. Mark the **Draw this component** check box.

6. If you use a PH10, Tesa, or continuous type wrists with a C joint, you need to verify that the joint angles are properly adjusted for visual purposes. Otherwise, PC-DMIS can't properly correlate the sensor's data to the machine position. If your probe is not rotated correctly about the joint, you can manually provide the extra rotation. To do this, right click on the component and change the **Default rotation angle about connection** value to reflect the needed rotation.

> **NOTE:** The probe file does not define the orientation of the sensor about the joint; it only defines the probe vector.

For additional information on defining probes, see the "Defining Hardware" section in the main PC-DMIS documentation.

Step 3: Define Setup Options for the Laser Sensor

> **NOTE:** If PC-DMIS is configured for laser sensor HP-L- 20.8 at startup, the system looks for the current mounted probe. If it is *not* the laser HP-L- 20.8 sensor and there is a tool rack present, the system assumes that the sensor is in the rack and switches on the warm-up power state. This ensures that the sensor is warmed and ready for measuring.

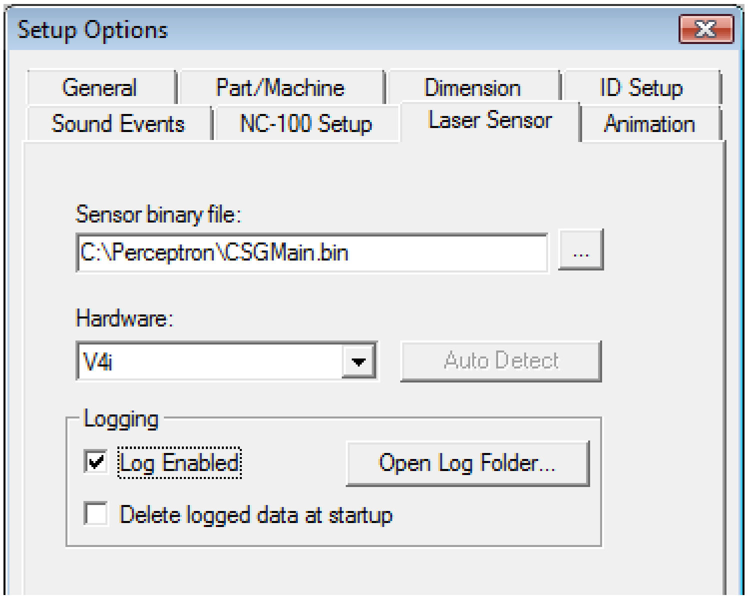
1. If you see the **Probe Utilities** dialog box from the previous step, close it down.
2. Select **Edit | Preferences | Setup** or press **F5** key to open the **Setup Options** dialog box
3. Select the **Laser Sensor** tab. The contents of this tab change based on the type of laser sensor that your license or portlock configuration specifies.
    - Perceptron Sensors
    - CMS Sensors
4. Follow the setup options instructions below for your laser sensor.

Registry Settings for Laser Sensors

A PH10 wrist can automatically switch between a contact probe and a Perceptron probe. These registry settings control that operation as well as the power on a laser sensor warm-up station:

- `PICSDifferentialSwitchBit`
- `WarmUpStationPowerBit`

Perceptron Sensors



Setup Options dialog box - Laser Sensor tab example pointing to the binary file for Perceptron sensors

Sensor Binary File - You can use the browse button (**...**) to browse to the location of the CSGMain.bin binary file. This binary file contains the sensor configuration that came with your probe. The process that installs the toolkit and drivers for your probe also installs this binary file.

Hardware list - You can specify the hardware and PC-DMIS remembers what options (Greysums, V5 Projectors, Flat Target Calibration, and so on) to allow or disallow even when you run PC-DMIS in offline mode. When offline, all the options for the selected hardware type are available for revision.

AutoDetect - This button checks the hardware attached to your machine. It verifies the hardware that you specified in the **Hardware** list is correct.

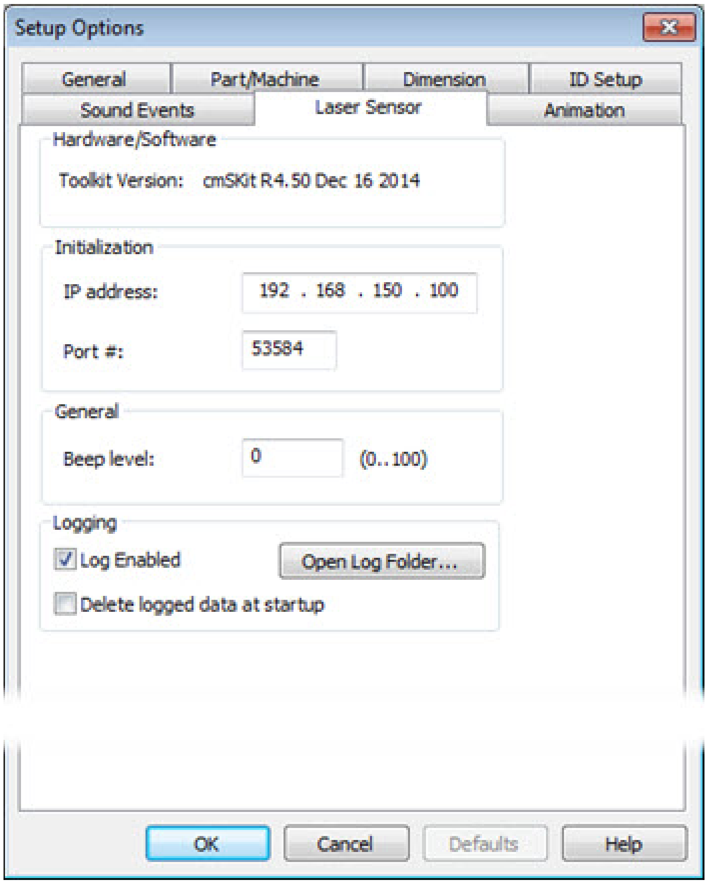
Logging area - You can use this area to generate text-based log files that contain communication results between PC-DMIS and the laser sensor when the measurement routine executes. The information it sends to the log files includes scans, nominals of calculated features, and so on. Technical support can use these files to resolve issues that involve your laser sensor.

- **Log Enabled** - This check box enables or disables data sent to the log files.
- **Open Log Folder** - This button opens up the folder that contains your log files.

> **EXAMPLE:** For example, for PC-DMIS 2017 R1, the folder content is located in C:\ProgramData\WAI\PC-DMIS\2017 R1\NCSensorsLogs\

- **Delete logged data at startup** - This check box deletes the logged data files from the log folder whenever you create a new measurement routine.

CMS Sensors



Setup Options dialog box - Laser Sensor tab example for CMS sensors

Hardware/Software - This area displays the current CMS toolkit version.

Initialization area - You can use the **IP address** and **Port #** boxes to define the IP address and the port number of the CMS controller.

General area - You can use the **Beep level** box to set the volume for beep sounds that come from the CMS controller. It can accept any value from 0 to 100. 0 turns off the volume completely.

Logging area - You can use this area to generate text-based log files that contain communication results between PC-DMIS and the laser sensor when the measurement routine executes. The information it sends to the log files includes scans, nominals of calculated features, and so on. Technical support can use these files to resolve issues that involve your laser sensor.

- **Log Enabled** - This check box enables or disables data sent to the log files.
- **Open Log Folder** - This button opens up the folder that contains your log files.

> **EXAMPLE:** For example, for PC-DMIS 2017 R1, the folder content is located in C:\ProgramData\WAI\PC-DMIS\2017 R1\NCSensorsLogs\

- **Delete logged data at startup** - This check box deletes the logged data files from the log folder whenever you create a new measurement routine.

The **Laser Sensor** tab also displays the installed CMS Toolkit version.

Step 4: Calibrate the Laser Probe

The calibration process described in this step may vary depending on the "Measure Laser Probe Options" and the type of installed interface. For detailed information on laser sensor calibration options, see the "Measure Laser Probe Options" topic.

Calibrating Perceptron Sensors

> **NOTE:** During calibration, PC-DMIS temporarily overrides your current exposure and gray sum values with the default exposure and gray sum values covered in the "Exposure and Gray Sum Settings During Calibration" topic. Once calibration finishes, the software restores your original values.

The following steps outline the procedure to calibrate your laser sensor for the first time:

1. Select **Insert | Hardware Definition | Probe** to open the **Probe Utilities** dialog box.
2. From the **Active Tip List** box, select the tip that you defined in Step 3
3. Click **Measure** to open the **Measure Laser Probe** dialog box (for information on this dialog box, see "Measure Laser Probe").
4. From **Type of Calibration Operation**, select one of the options. Then for Perceptron sensors, select **Offset**.
5. Define other calibration options as needed: **Motion** type, **Move Speed**, **Parameter Sets** and **Calibration Tool**.

> **IMPORTANT:** If you use a multi-sensor CMM with both a contact probe and a laser probe, make sure a calibrated contact probe first locates the sphere location for the laser calibration tool. This correlates the laser sensor measurement data with the contact probe calibration.

6. Click **Measure** to begin the calibration procedure. Follow any on-screen instructions. The first several prompts that you see are identical to the setup procedure for touch-trigger probes.

> **NOTE:** If you use the **MAN** or **MAN + DCC** motion options, or if you answer **Yes** to the message “Has the sphere moved”, you need to bisect the qualification sphere. For information, see "Bisecting the Calibration Sphere". Once you do an Offset calibration, the software no longer asks you to bisect the sphere unless you answer yes to the message “Has the sphere moved”.

> **HINT:** Certain sensor tip angles may cause the laser beam to fall on a portion of the calibration tool's stem. In some cases, the standard deviation for the sensor calibration of those tips exceeds the expected amount. In those cases, PC-DMIS displays a message to ask if you want to repeat the calibration of those tips. If you click **Yes**, the system uses the offsets and orientation determined by the first measurement rather than the theoretical values. This results in a clipping around the target that is more accurate during this re-calibration.

7. Once execution finishes, PC-DMIS returns to learn mode and shows the **Probe Utilities** dialog box.
8. Once the sensor calibration finishes, PC-DMIS shows the **Probe Utilities** dialog box.
9. If needed, click **Add Angles** to define any other tip angles that you need to calibrate.
10. From the **Active Tip List** box, select the tips that you want to calibrate. The initial tip calibration only finds offset information for the sensor configuration.
11. Click **Measure** to open the **Measure Laser Probe** dialog box. If you don't select any angles, the software asks if you want to calibrate all the tips.
12. From the **Measure Laser Probe** dialog box, select the **Tips** option.
13. For **Calibration Tool**, select the same tool that you used earlier.
14. Click **Measure** to begin the tip calibration. Once calibration finishes, PC-DMIS shows the **Probe Utilities** dialog box.

> **NOTE:** PC-DMIS stores Offsets of each axis for Perceptron sensors in the registry as `HotSpotErrorEstimateX`, `HotSpotErrorEstimateY`, and `HotSpotErrorEstimateZ`. For details, see "`HotSpotErrorEstimateXYZ`" in the PC-DMIS Settings Editor documentation.

> Once you perform either **Offsets** or **Sensor** calibration, depending on the sensor type, only steps 8 through 15 need to be performed on any new probe file which utilize the same sensor and CMM.

Calibrating Portable CMS Laser Sensors

The following steps outline the procedure to use to calibrate a portable laser CMS sensor using a planar artifact:

1. From the **Probe Utilities** dialog box, click **Measure** to open the **Measure Laser Probe** dialog box. For information on this dialog box, see "Measure Laser Probe Options".
2. Select the appropriate sensor mode. The default is **Zoom2A**.
3. Place the planar artifact in a convenient location for the arm to measure.
4. Click **Measure** to begin the calibration procedure. Follow any on-screen instructions.
5. The calibration procedure requires that you take 17 laser stripes on the planar artifact in different positions and orientations with respect to the planar artifact. To help you visualize where to take the stripe, the system draws a yellow target line on the **Laser View** tab of the Graphic Display window.

Calibrating DCC CMS Laser Sensors

The calibration process described in this step may vary depending on the laser sensor options and the type of installed interface. Refer to the "Measure Laser Probe Options" topic for detailed information on calibration options.

The following steps outline the procedure to calibrate your laser sensor for the first time:

1. Select **Insert | Hardware Definition | Probe** to open the **Probe Utilities** dialog box.
2. From the **Active Tip List** box, select the tip that you defined in Step 3.
3. Click **Measure** to open the **Measure Laser Probe** dialog box (for information on this dialog box, see "Measure Laser Probe").
4. Select the appropriate sensor mode. The default is **Zoom2A**.
5. Define other calibration options as needed: **Motion** type, **Move Speed**, **Parameter Sets** and **Calibration Tool**.

   > **IMPORTANT:** If you use a multi-sensor CMM with both a contact probe and a laser probe, make sure a calibrated contact probe first locates the sphere

location for the laser calibration tool. This correlates the laser sensor measurement data with the contact probe calibration.

6. Click **Measure** to begin the calibration procedure. Follow any on-screen instructions. The first several prompts that you see are identical to the setup procedure for touch-trigger probes.

NOTE: If you use the **MAN** or **MAN + DCC** motion options, or if you answer **Yes** to the message “Has the sphere moved”, you need to bisect the qualification sphere. For information, see "Bisecting the Calibration Sphere". Once you do an Offset calibration, the software no longer asks you to bisect the sphere unless you answer yes to the message “Has the sphere moved”.

7. Once execution finishes, PC-DMIS returns to learn mode and shows the **Probe Utilities** dialog box.
8. If needed, click **Add Angles** to define any other tip angles that you need to calibrate.
9. From the **Active Tip List** box, select the tips that you want to calibrate. The initial tip calibration only finds offset information for the sensor configuration.
10. Click **Measure** to open the **Measure Laser Probe** dialog box. If you don't select any angles, the software asks if you want to calibrate all the tips.
11. From the **Measure Laser Probe** dialog box, select the appropriate sensor mode. The default is **Zoom2A**.
12. Select the **Tips** option.
13. For **Calibration Tool**, select the same tool that you used earlier.
14. Click **Measure** to begin the tip calibration. Once calibration finishes, PC-DMIS shows the **Probe Utilities** dialog box.

HINT: Certain sensor tip angles may cause the laser beam to fall on a portion of the calibration tool's stem. In some cases, the standard deviation for the sensor calibration of those tips exceeds the expected amount. In those cases, PC-DMIS displays a message to ask if you want to repeat the calibration of those tips. If you click **Yes**, the system uses the offsets and orientation determined by the first measurement rather than the theoretical values. This results in a clipping around the target that is more accurate during this re-calibration.

A hardware configuration of a CMS laser sensor and an infinitely indexable wrist, such as the CW43L, has the ability to qualify infinite tip orientations. You can define the tip orientations by wrist angles A, B, and C through a Laser Wrist Map (LWM). You can create a LWM if you qualify a grid of tip orientations that cover the specified range of angles A, B, and C.

Once you create the LWM for a specific sensor, you can add new tips to the sensor and if those tips are within the angle range that you specified during map creation, they are automatically qualified and ready to measure with.

IMPORTANT: You need to recreate the LWM each time a component of the wrist changes (for example, when the CJoint changes). Also, refer to your hardware and vendor information for the appropriate times to map a wrist since this can change based on device construction and manufacturer recommendations.

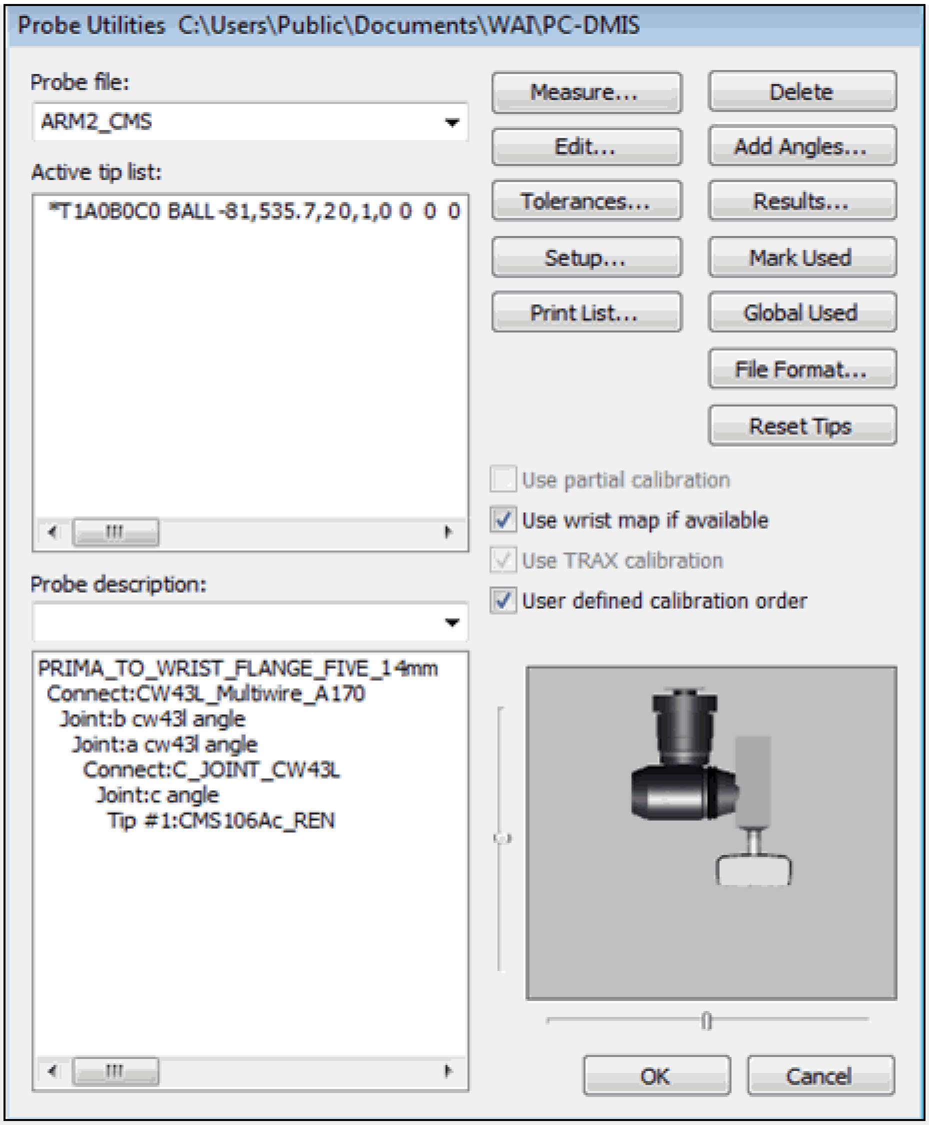
The following steps outline the procedure for mapping infinite wrist DCC CMS laser sensors:

1. Define the sensor:

    a. In the **Probe Utilities** dialog box, create a sensor as indicated below:

        - Infinitely indexable wrist, such as the CW43L
        - CJoint
        - CMS laser sensor

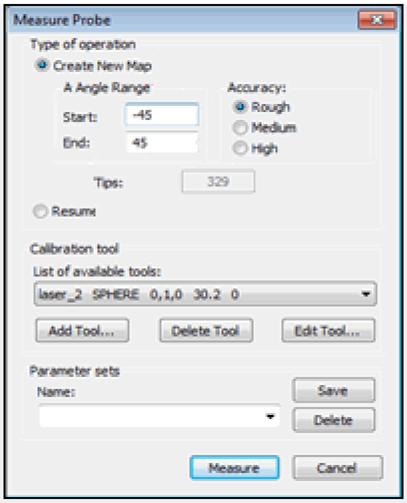
EXAMPLE: For example:



Sample Probe Utilities dialog box with a CMS laser sensor and an indexable wrist

b. Select the **Use wrist map if available** check box.
c. Click **Measure** to display the **Measure Probe** dialog box.

EXAMPLE: For example:



2. Create the map:

   a. From the **Measure Probe** dialog box, select the **Create New Map** option.
   b. For **A Angle Range**, type the desired **Start** and **End** values. These values define a range of angles that form a virtual cone. The map qualifies any tip orientations that fit in this virtual cone.

   NOTE: The B and C angles are always mapped within the full physical range (typically, -180 to +180 degrees).

   c. For **Accuracy**, select the desired option:

      - **Rough** - Step Angles: A ~40, B ~40, C ~40
      - **Medium** - Step Angles: A ~30, B ~30, C ~20
      - **High** - Step Angles: A ~20, B ~20, C ~10

   The **Tips** box displays the total number of tips measured to create the map.

   d. Click **Measure**.

      - PC-DMIS measures five sensor orientations around the sphere tool.
      - PC-DMIS measures all of the tips in the mapping grid.

Updating an Existing Map

Once you create the map, you can recover the correct qualification for all of the tips whenever a geometrical or thermal parameter of the Sensor - Wrist system changes. For example, after the sensor experiences a physical collision, or when the room temperature changes.

To recover the correct qualification:

1. From the **Measure Probe** dialog box, select the **Update the map** option.
2. Click **Measure**. PC-DMIS starts to remeasure the same five sensor orientations around the sphere tool as it measured during the map creation process.

Resuming Map Creation

If the process of creating a map is interrupted (because the machine powered down, you were interrupted, or some math calibration errors occurred, for example), a **Resume** option appears in the **Measure Probe** dialog box. You can use this option to continue creating the map.

To resume the process of creating a map:

1. Select the **Resume** option in the **Measure Probe** dialog box. PC-DMIS automatically calculates which tips are still missing in the current map and creates a list of the missing tips to be measured.

> **NOTE:** You are not able to use the **Resume** option again until the map is successfully completed.

2. Click **Measure**. PC-DMIS starts to measure the tips necessary to complete the map.

Defining Parameter Sets for Map Creation

You can define a parameter set to create a map. You can also use the `AUTOCALIBRATE` command within a measurement routine to update a map.

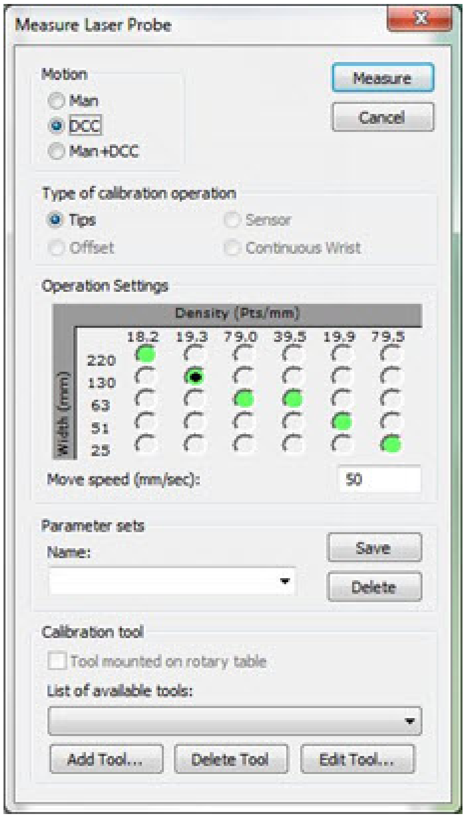
To define a parameter set:

1. In the **Measure Probe** dialog box, select and type the desired values.
2. In the **Name** box, type a name for the parameter set.
3. Click **Save**.
4. To close the dialog box, click **Cancel**.

For more information about parameter sets and using the `AUTOCALIBRATE` command, refer to "Dual Arms with Wrists Calibration Example" in the PC-DMIS Core documentation.

Measure Laser Probe Options

The options on the **Measure Laser Probe** dialog box determine the procedure that the software uses for the laser sensor calibration. To access this dialog box, from the **Probe Utilities** dialog box (**Insert | Hardware Definition | Probe**), click **Measure**.



Measure Laser Probe dialog box

Change the following options as needed or as directed in "Step 4: Calibrate the Laser Sensor".

Motion

- **Man** - This requires that you manually position the arm in several different locations that bisect the calibration tool. This varies based on the sensor manufacturer. This is the only available **Motion** option for arm machines.
- **DCC** - Use this mode when the laser sensor has accurate offsets provided by the sensor manufacturer or if you have already run the calibrate "offset" routine. This moves the machine through a series of positions as recommended by the sensor manufacturer. You are not required to position the sensor manually for each tip that is calibrated.

- **Man+DCC** - This mode is similar to DCC except that you must position the sensor over the sphere in order to begin the calibration sequence for each of the tips to calibrate. The software prompts you to position the sphere at the beginning of the calibration process.

Type of calibration operation

NOTE: The options in this section are available based on the laser sensor. **Tips** works for all probes, **Offset** is only for Perceptron sensors.

- **Tips** - Use this option to do a standard calibration or all marked tips for your laser sensor.
- **Offset** - Use this option to estimate the laser sensor offset for Perceptron laser sensor types. You only need offset calibrations to position the machine correctly to calibrate tips. If you skip this step, the probe may miss the sphere during tip calibration.

HINT: When calibrating Perceptron sensors for the first time:

1. With the Offset option, calibrate a single tip.
2. With the Tips option, calibrate the first tip angle and any other tip angles.

For more details, see "Step 4: Calibrate the Laser Sensor".

Operation Settings

The items that appear in this area vary based on the laser sensor type.

- **Sensor states** - As with the "Scan Zoom States (for CMS Sensors)" topic, this collection of option buttons appears only for CMS sensors. It lets you select a predefined sensor state. Each state has a specific combination of sensor frequency, data density, and Field of View (FOV) width.
- **Move Speed [%]** - Determines the percentage of the maximum machine speed that the software uses during the calibration process.

Parameter sets

Parameter sets allow you to create, save, and use saved sets for your laser sensor. This information is saved with the probe file and it includes the settings for your laser sensor.

To create your own named parameter sets:

1. Modify any parameters on the **Measure Laser Probe** dialog box.
2. From the **Parameter Sets** area, in the **Name** box, type a name for the new parameter set, and click **Save**. To delete a saved parameter set, select it and click **Delete**.

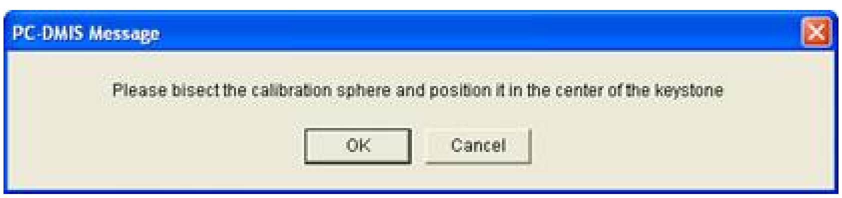
Calibration tool

Select the appropriate calibration tool. If this is your first calibration, you need to click **Add Tool** to first define the calibration tool. For specific information on defining a qualification tool, see the "Defining Hardware" chapter in the main PC-DMIS documentation.

IMPORTANT: Make sure you use the spherical qualification tool that comes with your laser sensor. The surface characteristics of this tool are designed for optimal scanning results. If you use a tool made by another manufacturer, it may produce inaccurate data.

Bisecting the Calibration Sphere

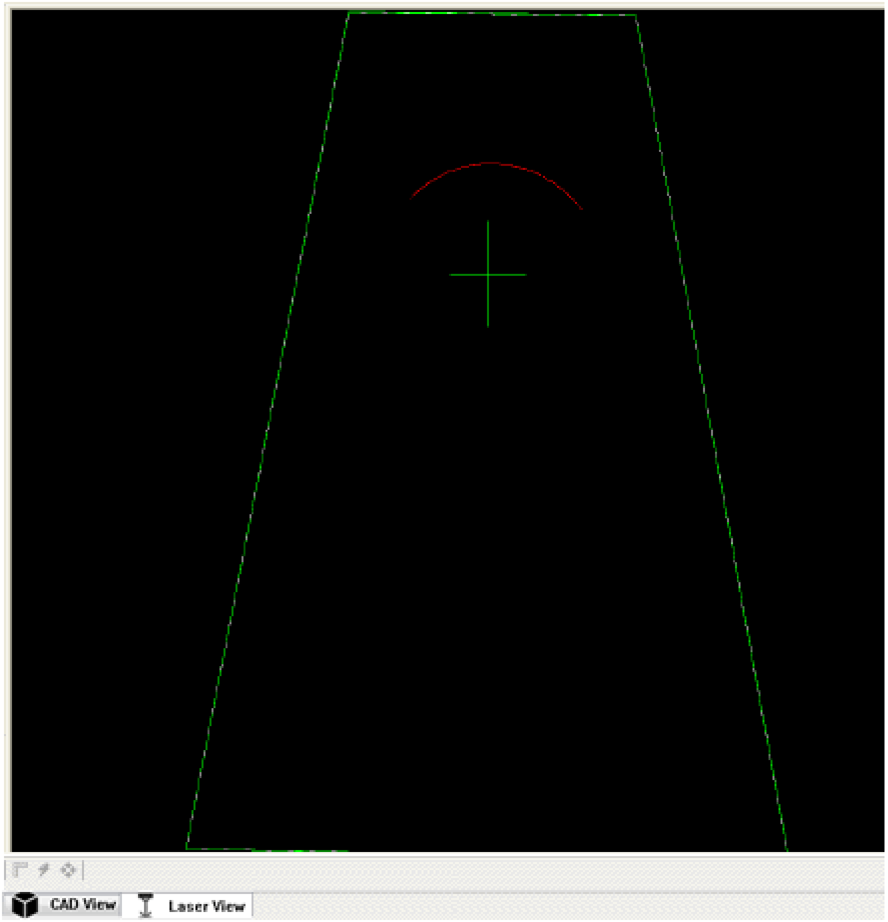
When you use either the MAN (Manual) or MAN + DCC Motion option you will be required to manually bisect the qualification sphere. This is also necessary if you have moved the sphere or do not know the location of the sphere. The calibration procedure will prompt you when you need to move the machine.



PC-DMIS Message

To Manually Bisect the Sphere:

1. Leave the PC-DMIS Message open.
2. Switch to the **Laser View** tab on the main **Graphic Display window**.
3. Click the **On/Off** button. This turns on the laser. A flashing red arc appears in the graphic area of the **Laser View** tab and a green crosshair. The red arc is where the laser hits the calibration sphere.
4. Center the cross hair inside the circular region formed by the arc by moving the machine with the jog box. The red arc moves as you move the machine. If you imagine that the flashing arc indicates the edge of a circle, the center point of this imaginary circle should optically align with the center of the cross hair.



Aligning the arc

5. Once you have aligned the arc, click the **On/Off** button again. This turns off the laser.
6. Click **OK** on the **PC-DMIS Message** to accept the change you made of aligning the arc. PC-DMIS stays in Execute mode and the laser sensor moves through a series of defined positions used to calibrate your tip.
7. At each position the laser beam hits the sphere in a stripe and the laser sensor collects the data from that stripe. The collected data and the corresponding machine position determine the sensor's mounting orientation on the machine.
8. Once execution finishes PC-DMIS returns to learn mode and shows the **Probe Utilities** dialog box.

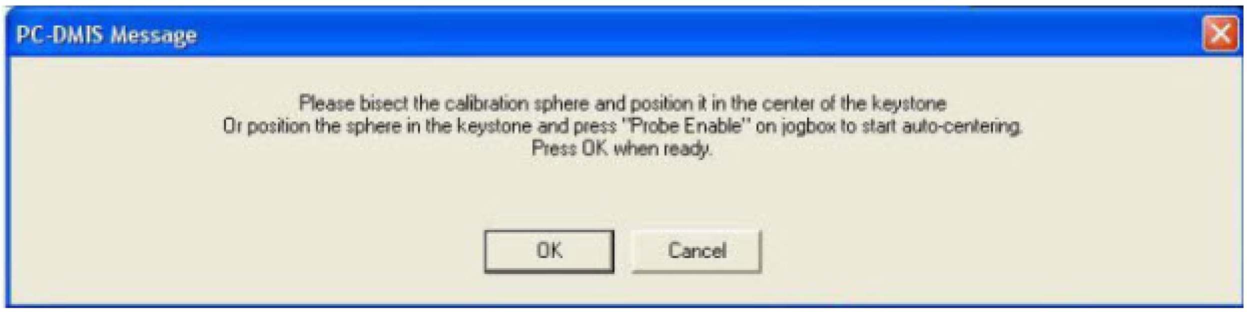
CMS Automatic Self-Centering of Tool Sphere

The CMS laser sensor provides automatic self-centering (bisecting) of the calibration tool sphere during calibration if you answer **Yes** to the question, "Has the sphere moved?". From the Graphic Display window, click the **Laser View** tab. You can drive the laser sensor to the center of the sphere.

You have two possibilities at this point:

- You can manually bisecting the sphere and bring it in the center of the Keystone and then press **OK** to start the laser calibration.
- Display a portion of the calibration sphere in the Laser View and then press the **Probe Enable** button to automatically center the sphere. When completed, press the **OK** button to complete the laser calibration.

The PC-DMIS Message dialog appears as soon as PC-DMIS determines that the calibration sphere has been moved.



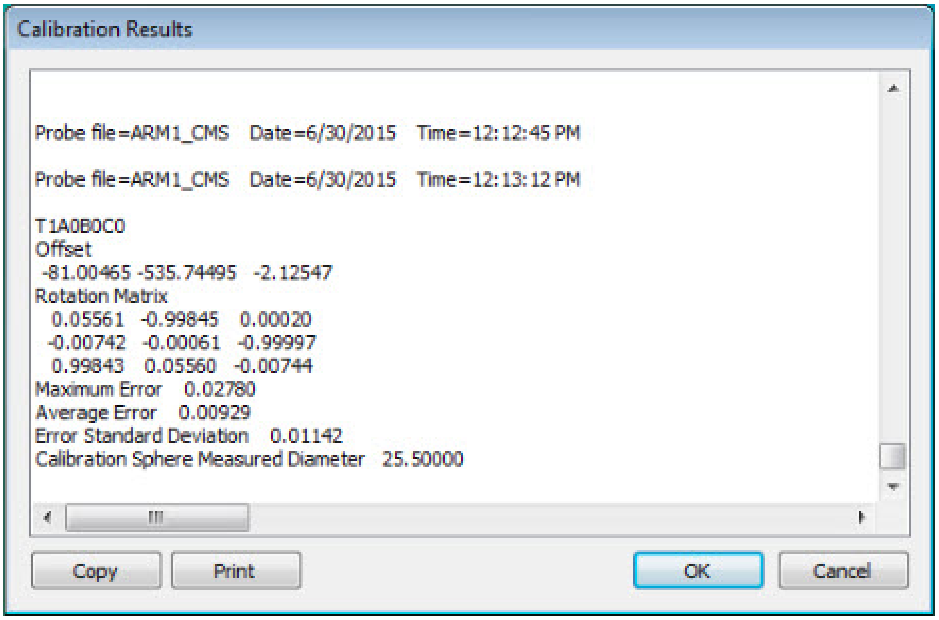
Follow the instructions as described in the message box.

Press the **OK** button when done.

HINT: For convenience, during the auto-centering procedure, the laser sensor alignment stripe appears in yellow.

Step 5: Check the Calibration Results

From the **Probe Utilities** dialog box, click on the **Results** button to show the **Calibration Results** dialog box.



Calibration Results

PC-DMIS records several things from the calibration in this dialog box. Take a look at the maximum, average, and standard deviation values.

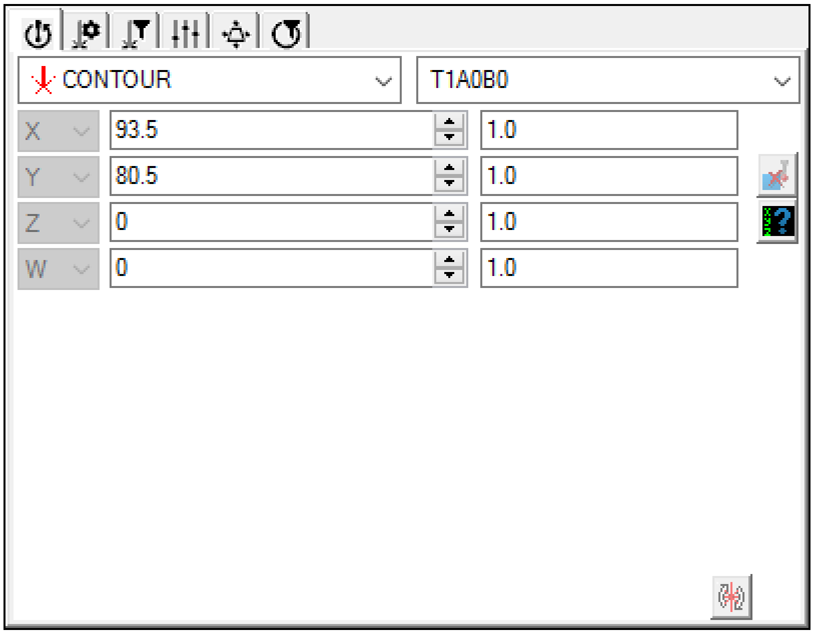
The maximum should be somewhere between 20 to 100 microns. The average and the standard deviation should be around 20 microns.

If things look correct, click the **OK** button to close the **Calibration Results** dialog box. You have these options:

- To paste the report into a different application (such as Microsoft Word, NotePad and so on), click **Copy** open the desired application press CTRL + V to paste it.
- To send the report to a printer, click **Print**.

This finishes the setup and calibration process for your laser sensor. You can now use all the laser-related functionality.

Using the Probe Toolbox in PC-DMIS Laser



Probe Toolbox with the laser sensor related tabs

The **View | Probe Toolbox** menu option displays the **Probe Toolbox**. The **Probe Toolbox** contains various laser sensor parameters that you can use to acquire the data points needed by a measurement routine.

> **IMPORTANT:** Your license or portlock contain the Laser option, and you must be working with a supported laser sensor in order to access the laser-related tabs in the **Probe Toolbox**.

The **Probe Toolbox** contains the laser parameters within these tabs:

For Portable Configurations:

- Laser Scan Properties *^+!
- Laser Filtering Properties *+!
- Laser Pixel Locator Properties *
- Feature Extraction ^!

For CMM Configurations:

Position Probe

Laser Scan Properties

Laser Filtering Properties

Laser Pixel Locator Properties

Laser Clipping Region Properties

Feature Extraction

CWS Parameter

NOTE: The list above shows all possible **Probe Toolbox** tabs. The tabs available depend on the sensor that you have on your system. If capabilities for a tab do not apply to your specific sensor, then that tab is not available.

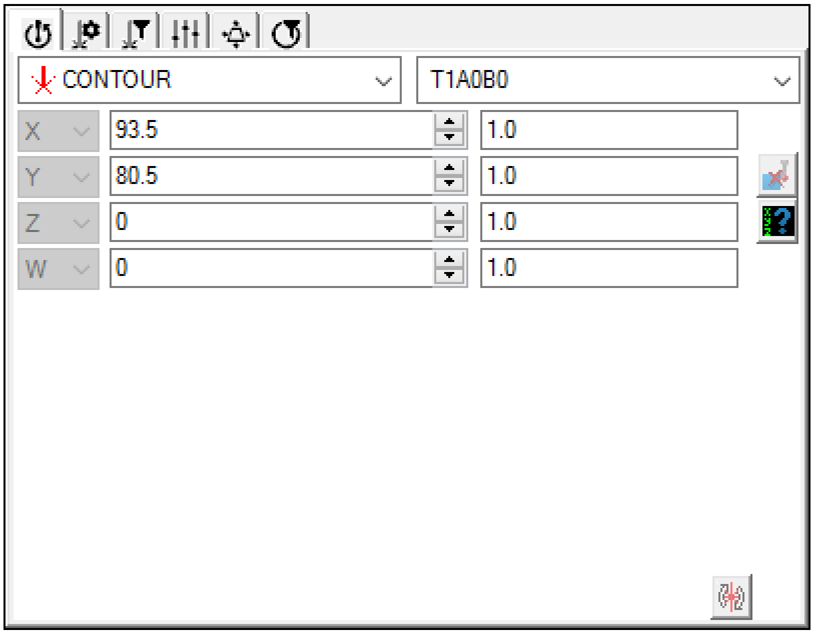
***** *For Perceptron probes*, these tabs are visible when you close the **Auto Feature** dialog box.

^ *For Perceptron probes*, these tabs are visible when you open the **Auto Feature** dialog box.

+ *For CMS probes*, these tabs are visible when you close the **Auto Feature** dialog box is closed.

! For CMS probes, these tabs are visible when you open the **Auto Feature** dialog box.

Laser Probe Toolbox: Position Probe tab



Probe Toolbox - Position Probe tab

The **Position Probe** tab of the **Probe Toolbox** (**View | Other Windows | Probe Toolbox**) allows you to select the current probe file and tip and define the current probe location in the active alignment coordinates. You can double click the X, Y or Z values to edit them.

Warning: When you edit the current probe location, the machine moves to the new coordinate without notice. To avoid injury, stay clear of the laser and the machine. To avoid hardware damage, run the machine at a slower speed.

If you don't see any information in the **Probes** and **Probe Tips** lists of the **Probe Toolbox**, you need to first define a probe. For information on how to define a probe, see the "Defining Hardware" section in the main PC-DMIS documentation.

NOTE: While you can use this tab with all probe types (contact, laser, or optical), this document only covers PC-DMIS Laser related items. For information about the toolbox as it relates to probes in general, see the "Using the Probe Toolbox" topic in the main PC-DMIS documentation.

To Position Your Laser Sensor

You can use the **Position Probe** tab of the **Probe Toolbox** (**View | Other Windows | Probe Toolbox**) to position your laser sensor. This tab contains sets of values in two columns.

Left Column: The X, Y, Z values. They show the laser sensor's current position. You can click up and down arrows to change the value in the **XYZ Probe Position** box  for an axis. This moves your laser sensor in real time by the increment value on the right.

Right Column: The increment values. These specify how much to increase or decrease the XYZ Probe Position box for each axis when you click the up and down arrows on the left column.

Alternately, you can type the XYZ values in the left column and press Enter to move your laser sensor to a predefined position.

Controls for Position Probe Tab

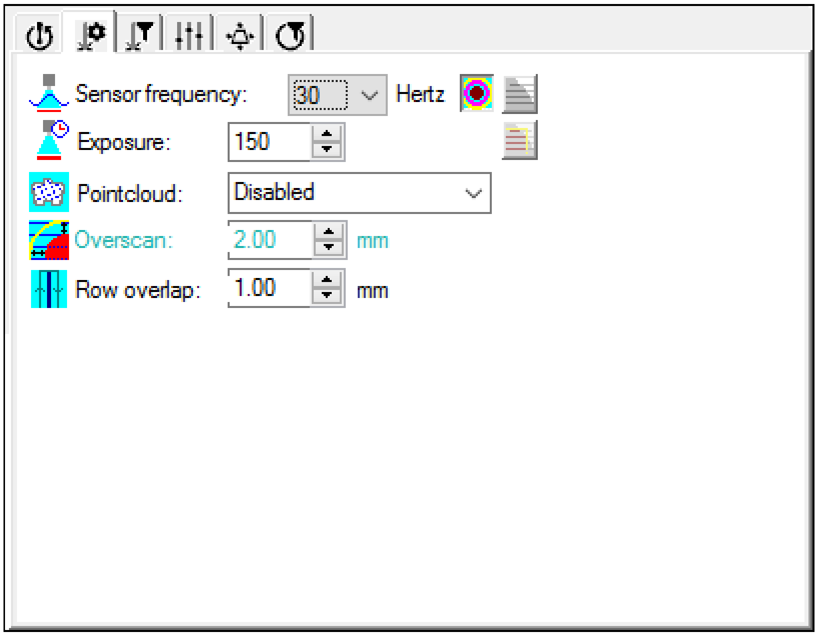
These describe the toggle buttons on **Position Probe** tab of the **Probe Toolbox** (**View | Other Windows | Probe Toolbox**):

 **Probe Readout toggle** - This toggle button shows or hides the Probe Readout window. You can easily resize or relocate this window. Most information on the Probe Readout window is the same for all probe types and is already discussed in the "Using the Probe Readout Window“ topic of the "Using Other Windows, Editors, and Tools" section in the main PC-DMIS documentation.

 **Toggle Laser On / Off** - This toggle button turns the laser on and off. It is only available for laser probes.

 **Initialize Probe** - This button starts or initializes the laser. You can't do anything with the laser until it is initialized. This takes about 15 seconds. (This button appears on this tab for DCC configurations.)

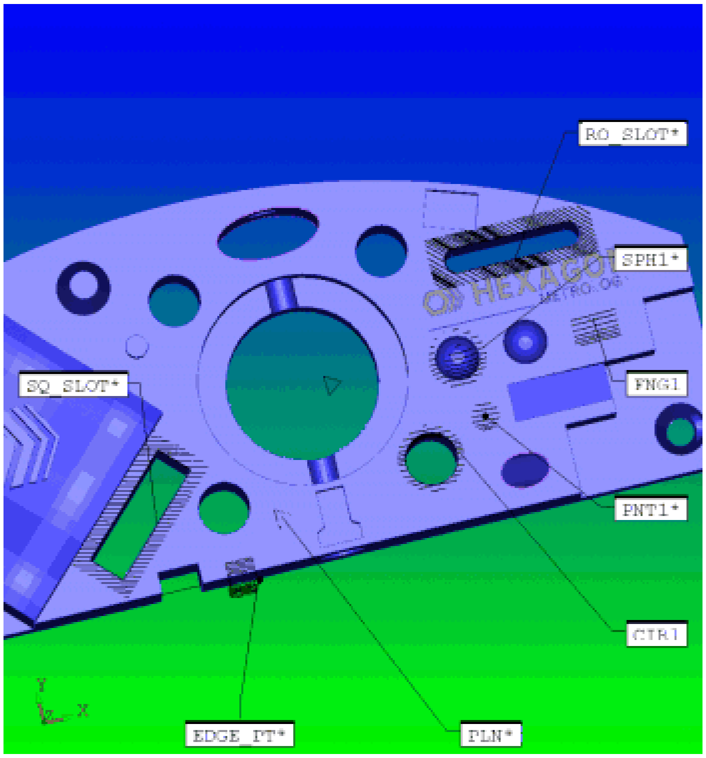
Laser Probe Toolbox: Laser Scan Properties tab



Probe Toolbox - Laser Scan Properties tab

The **Laser Scan Properties** tab defines how data from the scan will be acquired and whether or not scan lines and feature visualizations be displayed in the Graphic Display window.

 **Show/Hide Stripes** - This toggles the display of the laser stripes on the part model. Clicking this button causes the laser scan stripes to be displayed in real-time. PC-DMIS limits how the stripes are displayed in the Graphic Display window to the distance of the feature nominal plus the Overscan value. The **Overscan** value controls how much of the stripe is clipped and visible to the user. The graphic below gives an example of how these stripes are displayed.

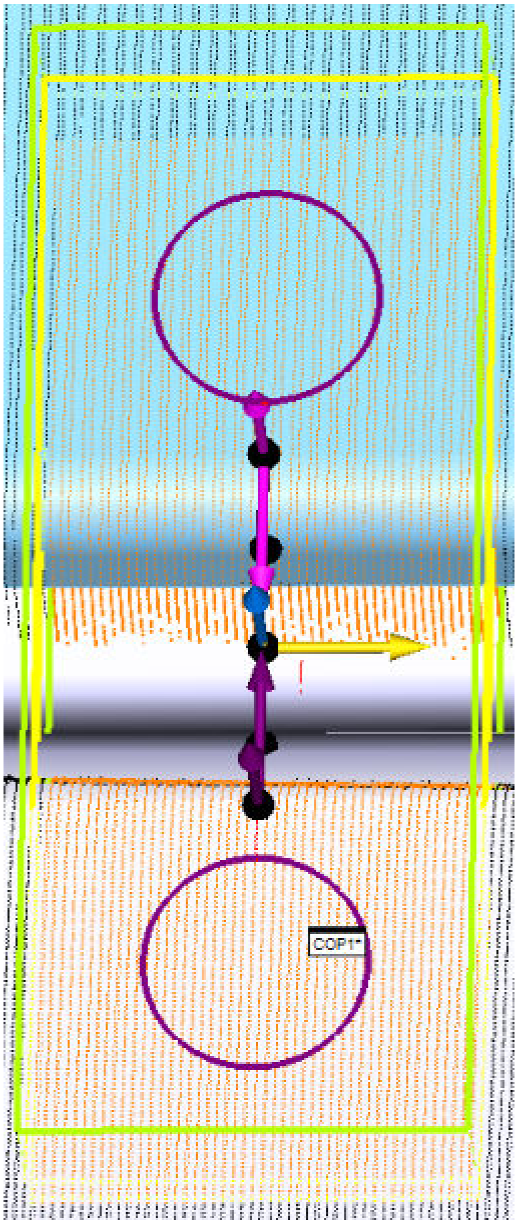


Scan Features showing stripes

Sound ON/OFF - This turns the sound on or off. See "Using Sound Events".

Visualization Tools ON/OFF - This toggles the display of the colored visualization tools. See "Understanding the Visualization Tools" for more information.

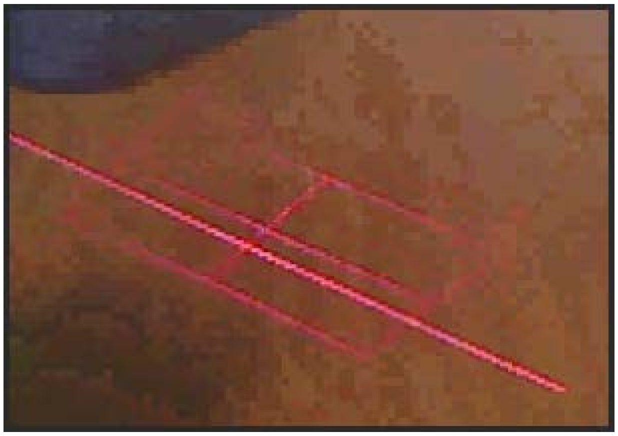
Show/Hide Segregated Points - This toggles the *display of those points* that will be passed to the feature extractor engine based on the current settings.



Showing segregated points inside a sample Flush and Gap feature

Initialize Probe - This button starts or initializes the laser. You can't do anything with the laser until it is initialized. This takes about 15 seconds. (This button appears on this tab for portable configurations.)

Projector: This is only available for V5 Perceptron probes on manual arms. Clicking this button turns on a projected *grid of red light* that will shine on the part. This acts like the cross hairs on a target. As you move your probe toward or away from your part, your probe's laser scan line moves through this target. For optimal results, your laser's scan line should line up with the center line of this target. This essentially serves the same purpose as the scan line indicator, helping you to keep the probe at the optimal height when measuring the part. Since this only functions in manual applications, this icon is disabled if using the **Probe Toolbox** within the **Auto Feature** dialog box.

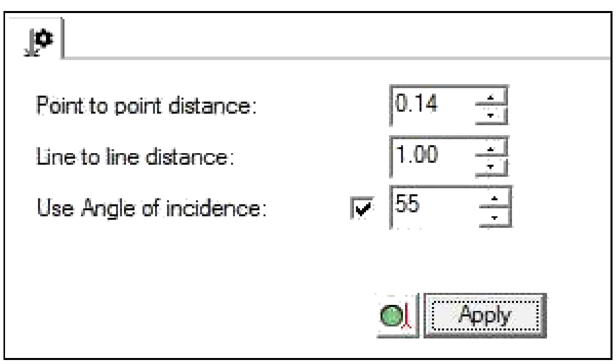


This actual picture of the Projector shows the rectangular grid-like projection of light. The brighter horizontal line is the laser's scan line.

 **AutoZoom ON/OFF** - This turns the laser AutoZoom functionality on or off. Whenever you start the scan, AutoZoom dynamically pans, zooms, rotates, and sizes the view containing the laser data in the Graphic Display window to show the incoming data.

Laser Scan Properties for a Leica T-Scan

For a portable Leica T-Scan probe, the **Laser Scan Properties** tab would contain these options:

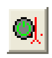


Probe Toolbox - Laser Scan Properties tab for Leica T-Scan

Point to point distance - This specifies the distance between two consecutive points in a scan line. Allowed values are between 0.035 mm and 10mm when using the up and down arrows.

Line to line distance - This specifies the distance between two consecutive scan lines. Allowed values are between 0 mm and 50mm when using the up and down arrows.

Use Angle of incidence - This specifies which maximum allowed angle will be used for scanning. This value helps avoid bad conditions while scanning (surface reflections, geometry, and so on). This angle is the angle between a ray and the surface normal vector. Allowed values are between 0 and 80 degrees when using the up and down arrows. If your mark the check box to the left of the box, PC-DMIS will send the angle value in the field. If you clear the check box, PC-DMIS will send a 90 degree angle to the dispatch interface. Typing a value of 90 degrees is the same as clearing the check box.

Initialize Scanner -  This icon starts the T-Collect software and initializes the scanner using the values defined in this tab.

Apply - This button applies the values defined in this tab without stopping the scanner.

NOTE: You can override the limitations with the up and down arrows or enter a value directly into any of the boxes. Invalid values however, are rejected by your machine and forced to a valid number.

Other Properties

Sensor Frequency

This parameter controls the internal sensor frequency of the probe. The value that is displayed is sensor pulses per second. For sensors with variable frequency capabilities, the higher the frequency, the more data you will get. It is important to understand that more data is not always better. With variable frequency scanners you should use a frequency in the middle of the supported range. This is a good balance between speed and accuracy.

Row Overlap

If the feature or patch scan is larger than the width of the scan line, multiple passes of the probe will be taken. In that case, this parameter controls how far each pass will overlap the previous pass. The default value is 1.0 mm.

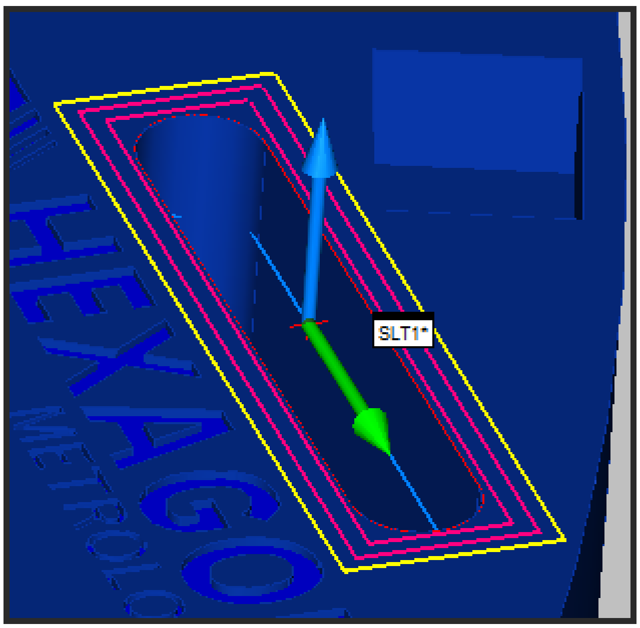
Overscan

For DCC systems, this parameter controls how far beyond the nominal feature's dimensions the probe will scan along both the major and minor axis of the feature. The default value is 2.0 mm. If you are measuring features whose actual location may vary significantly from their theoretical values, you will need to increase this value in order to ensure that PC-DMIS measures the entire feature.

In version 2010 and higher the **Overscan** value no longer does any sort of clipping of the data. Clipping is now handled by the new **Feature Based Clipping** area in the **Feature Extraction** tab. See the "Feature Based Clipping Parameters" topic.

For a DCC laser Cylinder or Cone feature, the **Overscan** value should be a negative value.

For a laser Stud feature (see the Laser Cylinder for stud information), the **Overscan** value should be a positive number.



Sample Slot Feature Showing the Overscan in Yellow

Exposure

This parameter controls the exposure of the sensor. The default value of 150 will work well for most parts, but for parts that absorb a lot of light (such as a black anodized surface), you may need to increase the value. If you're using a sensor that supports the Gray Sum pixel locator type, PC-DMIS will set the Exposure value to a material-specific value when you choose a material type from the **Material** list on the **Laser Pixel CG Locator Properties** tab of the Probe Toolbox.

The following table shows the available minimum and maximum exposure values for the supported Perceptron probes:

	Perceptron Laser Probes		
Normalized Exposure	V4i (Portable)	V4ix (DCC)	V5
Minimum Value:	32	1	1
Maximum Value:	627	627	1716
Default Value:	150	150	

Setting this to an inappropriate value may result in less accurate measurements.

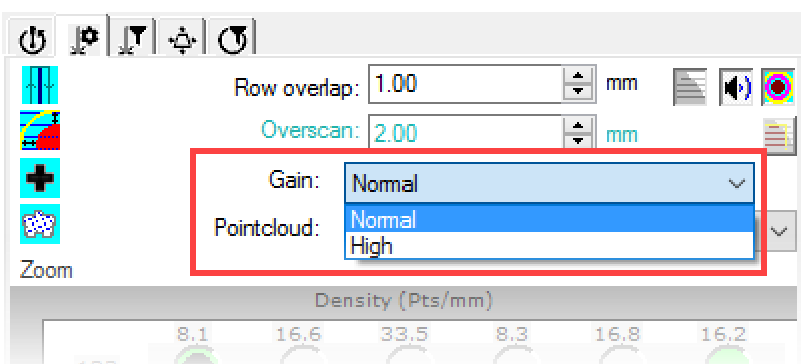
HINT: For Perceptron sensors, you can use the **Toggle AutoExposure** button on the **Laser View** tab to calculate the best exposure value for you. In addition, if you set the `AutoExposeWithLiveView` registry setting to TRUE, PC-DMIS will automatically set the exposure value in the probe toolbox to the best value every time you start the Laser View.

Point Cloud

This parameter defines the COP command from which the auto feature will be extracted. If "disabled" is selected, then the data from the scan will be stored internally by PC-DMIS. You can delete internal data if needed by using the **Operation | Laser Autofeatures** submenu. See "Clearing Auto Feature Scan Data".

HINT: The "disabled" option is only used with DCC laser scans.

Gain (for CMS Sensors)



Gain list

CMS sensors provide you with an additional list called **Gain** which is added onto the **Laser Scan Properties** tab of the **Probe Toolbox**. This lets you choose between these sensitivity modes:

CMS106 and CMS108 support **NORMAL** and **HIGH**. HP-L-20.8 supports **NORMAL**, **HIGH**, and **XHIGH**.

Sensitivity Modes

NORMAL sensitivity - You should use this default sensor mode on most normal parts. This mode sets the `QUALITY FILTER` toggle field in the Edit window's Command mode to `ON`, so that the Edit window shows the associated fields. This sensitivity mode also hides the **Quality Filter** icon.

HIGH sensitivity - The **HIGH** sensitivity mode becomes available for selection if you run PC-DMIS in online mode. You should only use the **HIGH** sensitivity mode if you are scanning a part comprised of a troublesome material where the **NORMAL** sensitivity mode returns poor data. For example, a part that absorbs too much light because it has glossy, dark, or black surfaces may require this type of mode. However, note that scanning a normal part in **HIGH** sensitivity mode may yield noisy data.

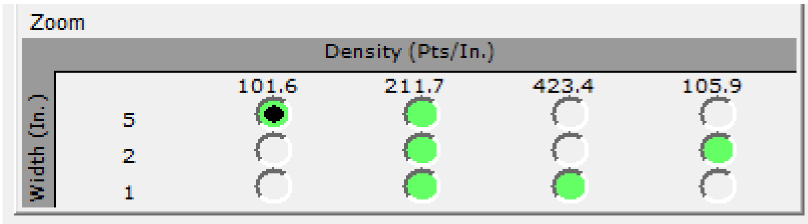
XHIGH (extra high) sensitivity - **XHIGH** is similar to **HIGH**. It provides an option for scanning materials that may be even more troublesome than those that can be handled using the **HIGH** option. If you don't get good results using **HIGH**, you can try using the **XHIGH** option. However, as with the **HIGH** option, if you scan a normal part in **XHIGH** mode, it may return even more noisy data.

In the **HIGH** and **XHIGH** modes, a **Quality Filter** icon appears next to the **Gain** list:

Quality Filter  - If you enable this mode, PC-DMIS filters low quality points including double reflections, poor quality data on edges, and outliers. If enabled, it sets the `QUALITY FILTER` toggle field to ON in the Edit window's Command mode to show the associated fields in the Edit window.

Scan Zoom States (for CMS Sensors)

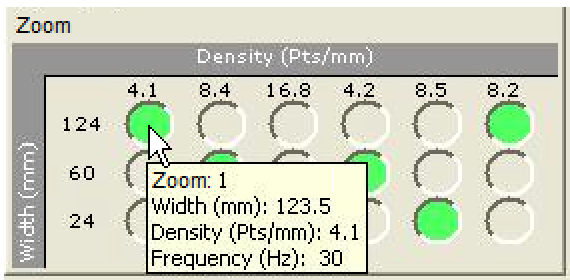
CMS sensors provide you with an additional area called **Zoom** which is added to the bottom of the **Laser Scan Properties** tab of the **Probe Toolbox**. This area tells the sensor to work in predefined zoom states, each state being comprised of a specific combination of sensor frequency, data density, and Field of View (FOV) width.



Sample Zoom area

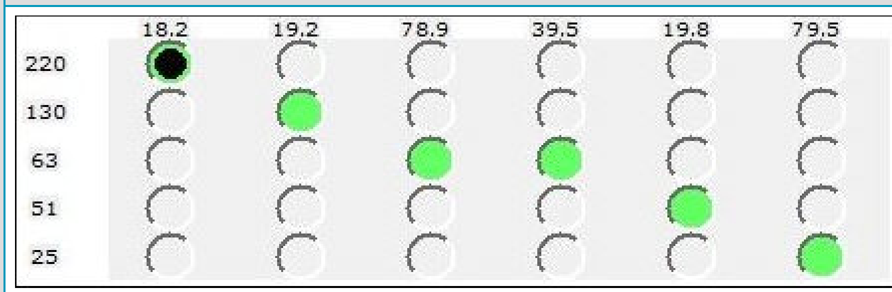
This area displays a table-like arrangement of option buttons arranged in columns and rows. Across the top, the "columns" show the data density. Along the side, the "rows" list the FOV width. You can only select proper combinations (those option buttons with a green background). Improper combinations are grayed out.

Hovering your mouse over any valid option button displays the selected scan mode information in a yellow tooltip.

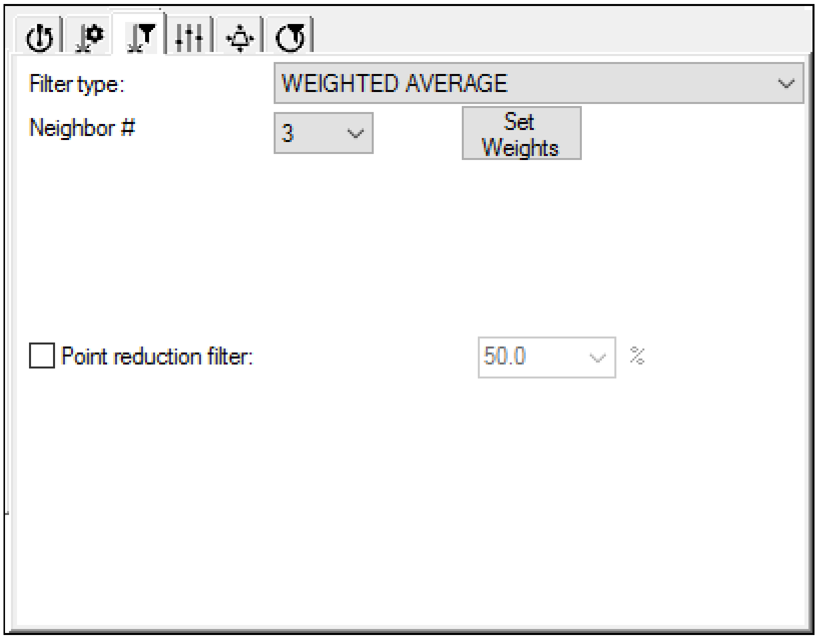


Sample ToolTip under the Mouse

Available Scan Zoom States for HP-L-20.8

Width (mm)	Density (Pts/mm)
	

Laser Probe Toolbox: Laser Filtering Properties tab



Probe Toolbox - Laser Filtering Properties tab

The **Filtering** tab is useful when you want to filter the data as it is collected.

NOTE: Scanning methods with a portable device using a Perceptron laser differ from DCC machines. If you open the **Auto Feature** dialog box and are using a portable device with a Perceptron laser, the **Laser Pixel CG Locator Properties** tab is hidden.

The following filtering options are available from the list:

Filter Type: Only Available for Perceptron Sensors

- **None** - Filtering will not take place if you select 'None'. This is the default setting.
- Long Line
- Median
- Weighted Average

Filter Type: Only Available for CMS Sensors

- Stripe

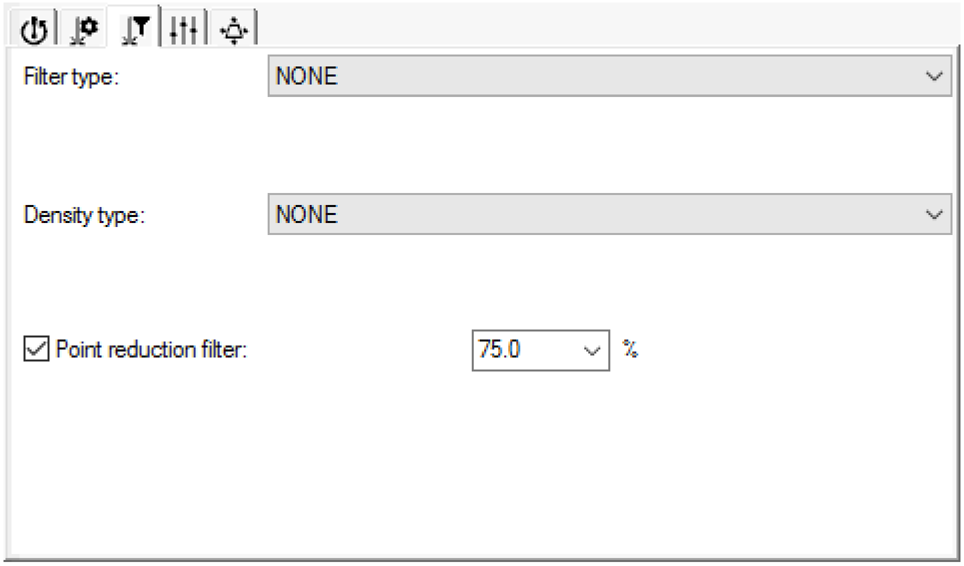
Density Type: Only Available for Perceptron Sensors

- **None** - Density filtering will not take place if you select **'None'**. This is the default setting.

- Intelligent Density Management (Contour V5 only)

NOTE: In PC-DMIS 2010 MR3 and later, the **Point** filter type for CMS and **Column Sampling Rate** for Perceptron have been combined in a generic **Point Reduction Filter** check box visible on all Filter types regardless of the laser sensor used.

Filter Type: None

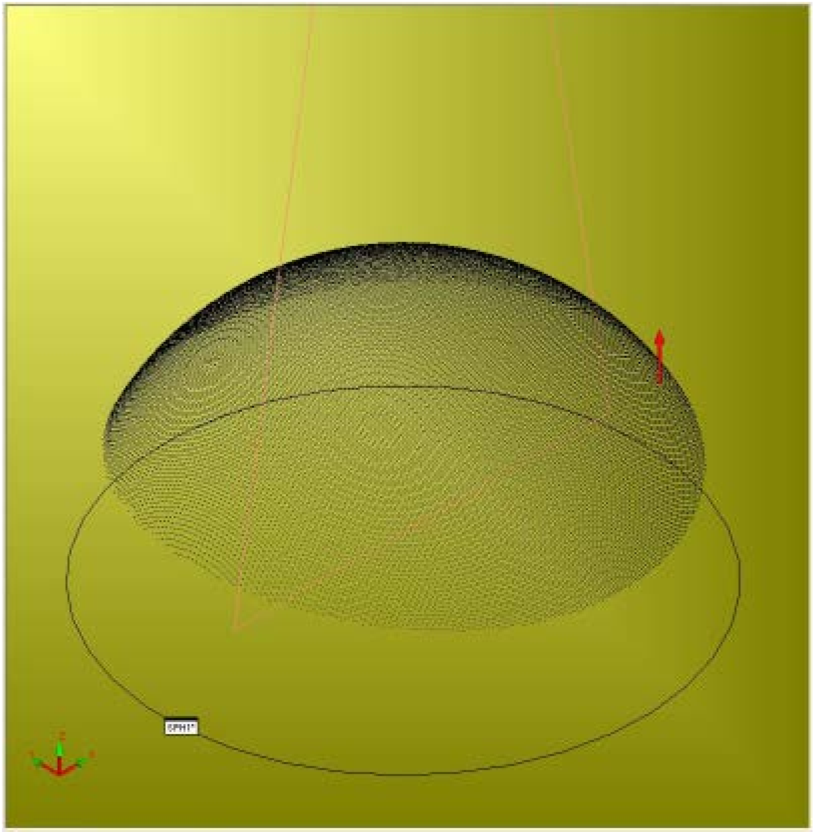


None Filter Type

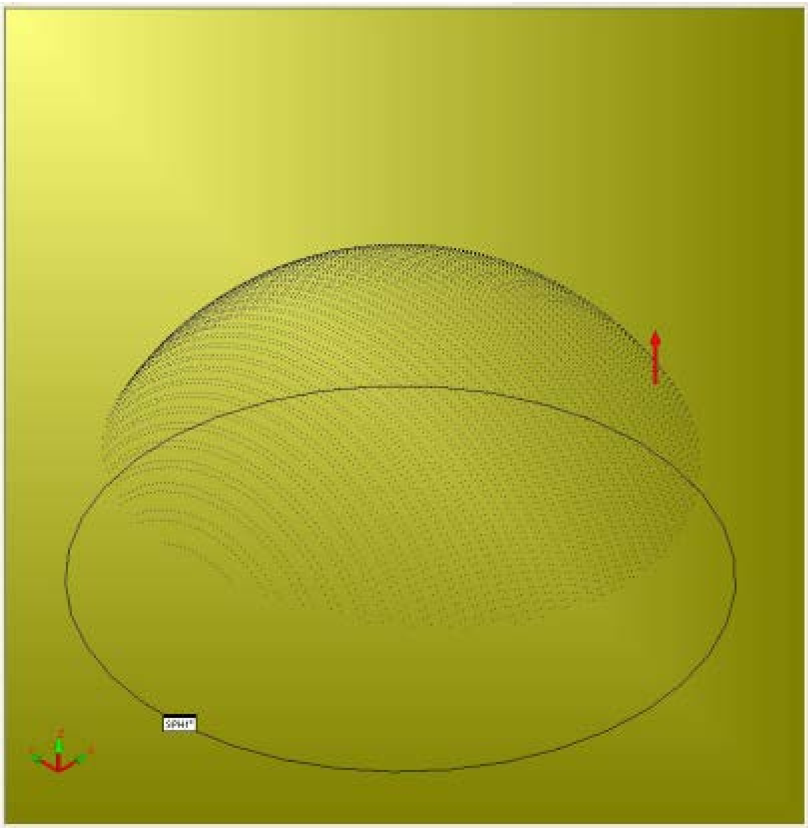
No initial filter is made. However, you do have the option of filtering by point reduction.

Point Reduction Filter: This check box determines whether or not PC-DMIS filters points along the scan line. If marked, you can select the desired percentage of total points to filter. If cleared, the complete data set is acquired without any filtering.

Example of Point Filtering Disabled

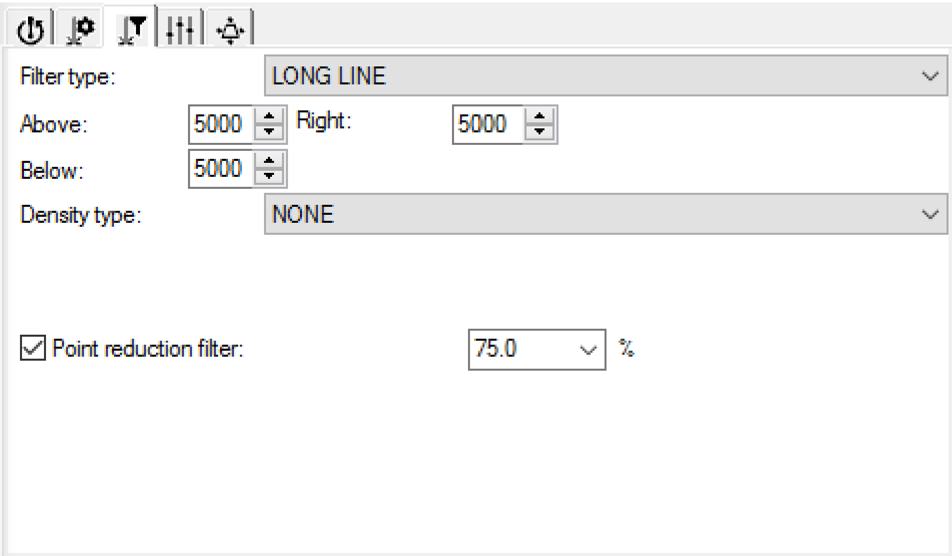


Example of Point Filtering of 50%



Filter Type: Long Line

> **NOTE:** This is only available for Perceptron sensors.



Long Line Filter Type

This filter is usually only used for measuring spheres and some cylinders.

The **Long Line** filter finds the longest continuous line or stripe of data in the image and rejects the rest of the data. The long line filter is also forced to be used during calibration. The laser stripe may be broken up due to the geometry of the part being measured. This filter finds the longest unbroken line. This is often used with sphere measurements. A section of the strip is considered continuous based on the following parameters:

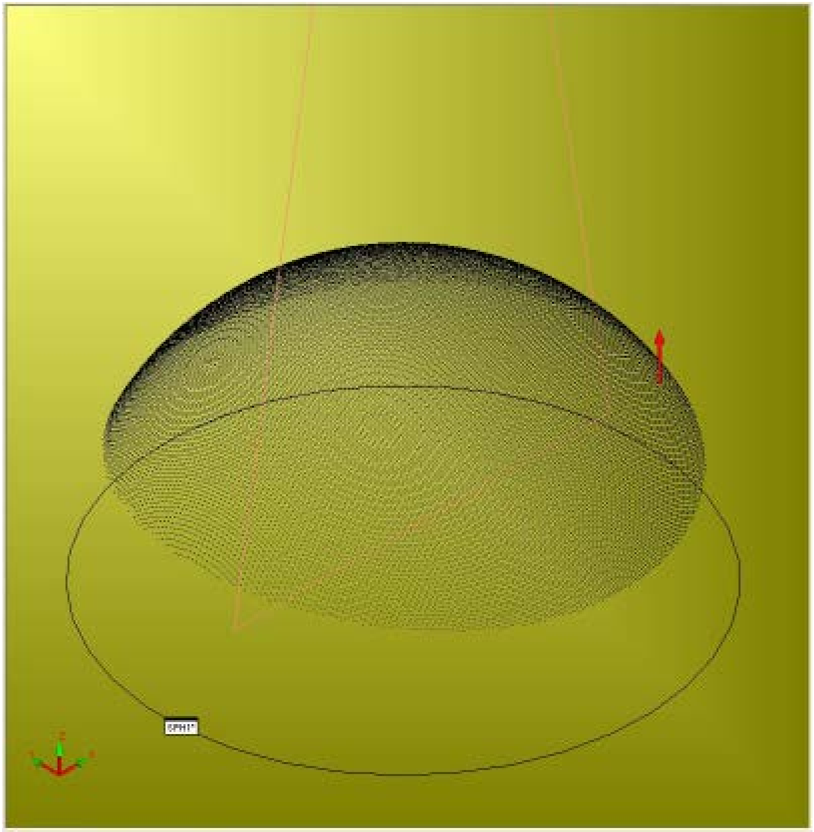
Above: This value determines the number of pixels in the image that the next pixel is allowed to rise and still be accepted as part of a continuous line. The value indicates the number of milli-pixels above the current pixel that the filter will still use.

Below: This value determines the number of pixels in the image that the next pixel is allowed to fall and still be accepted as part of a continuous line. The value indicates the number of milli-pixels below the current pixel that the filter will still use.

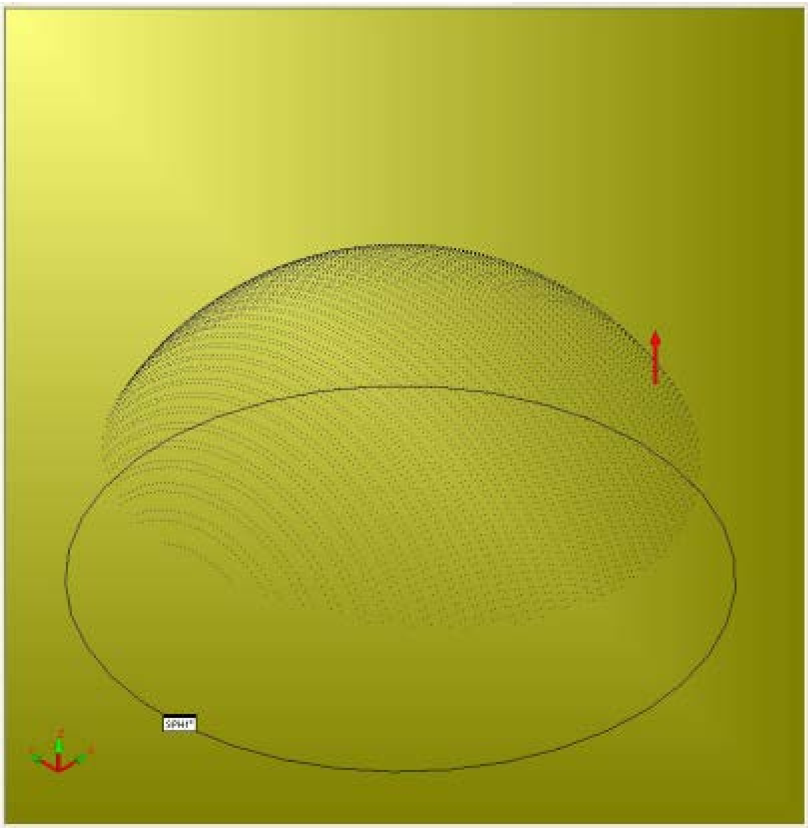
Right: This value determines the number of permissible missing milli-pixels to the right of the current pixel and is still considered a continuous line.

Point Reduction Filter: This check box determines whether or not PC-DMIS filters points along the scan line. If marked, you can select the desired percentage of total points to filter. If cleared, the complete data set is acquired without any filtering.

Example of Point Filtering Disabled

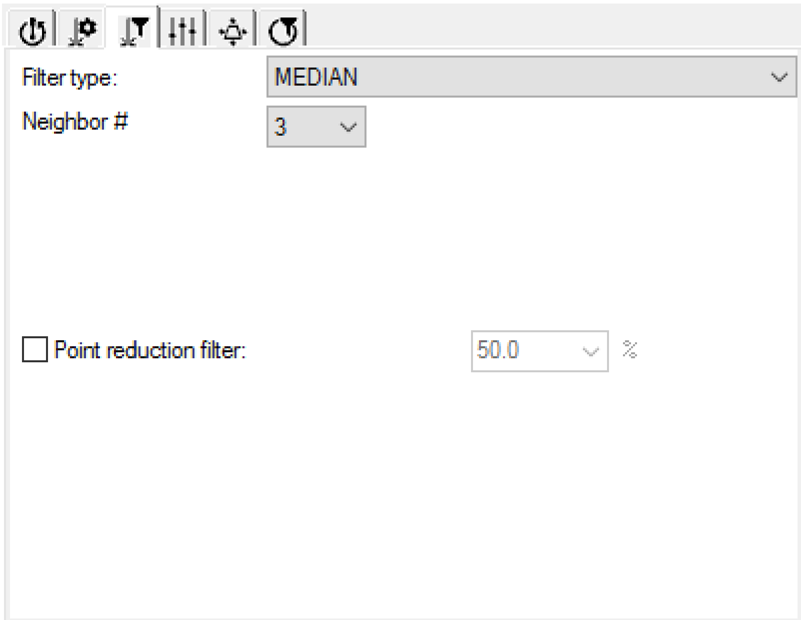


Example of Point Filtering of 50%



Filter Type: Median

NOTE: This is only available for Perceptron sensors.



Median Filter Type

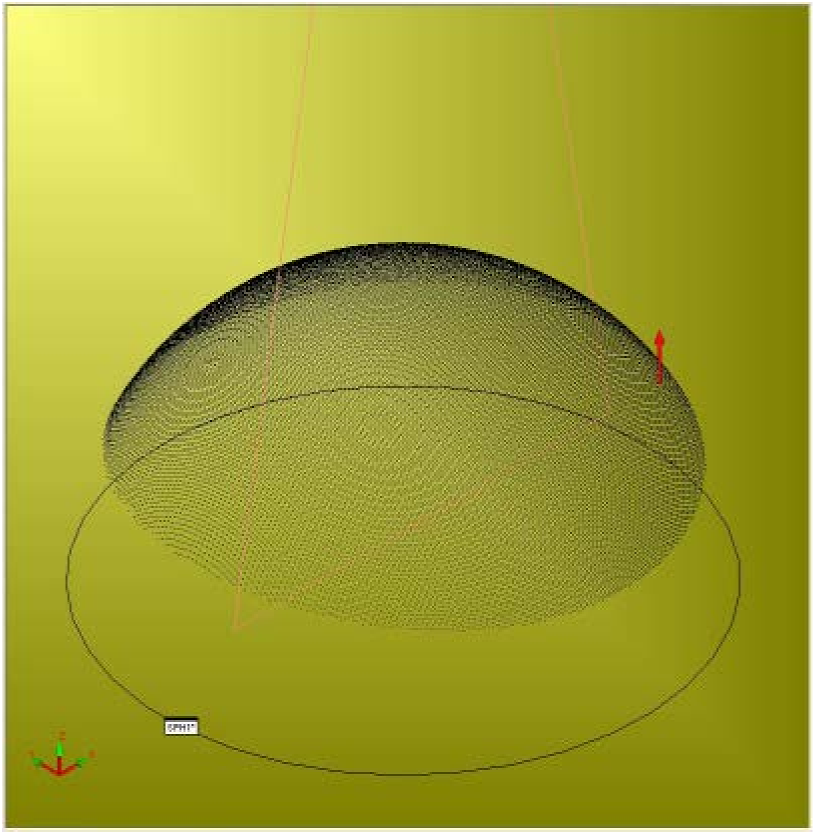
The **Median** filter smooths laser stripe data by computing a new location for each pixel. For each pixel in the stripe, the median filter takes the nearest neighboring pixels, computes the median, and uses the median for the new location of the pixel.

Neighbor #: This value determines the number of total neighboring pixels considered when calculating a new location of any given pixel in a single stripe.

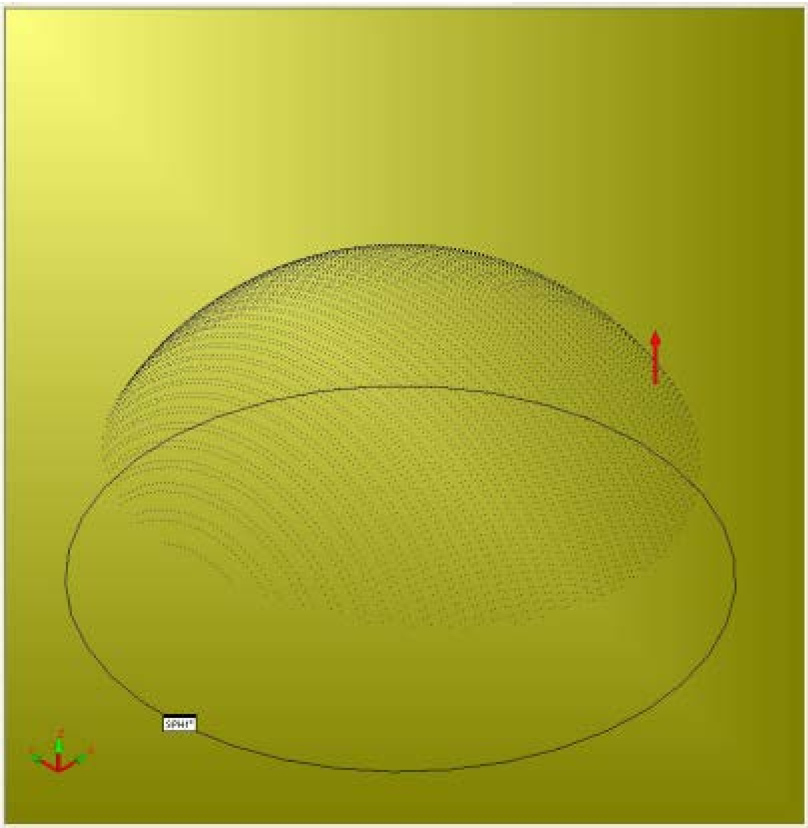
For example, if the number of neighbors is 9, then for each pixel in the stripe the filter will take four data points to the left and four data points to the right (for a total of 9 pixels, including the current one). It then computes the median, and uses it for the location of the current pixel.

Point Reduction Filter: This check box determines whether or not PC-DMIS filters points along the scan line. If marked, you can select the desired percentage of total points to filter. If cleared, the complete data set is acquired without any filtering.

Example of Point Filtering Disabled

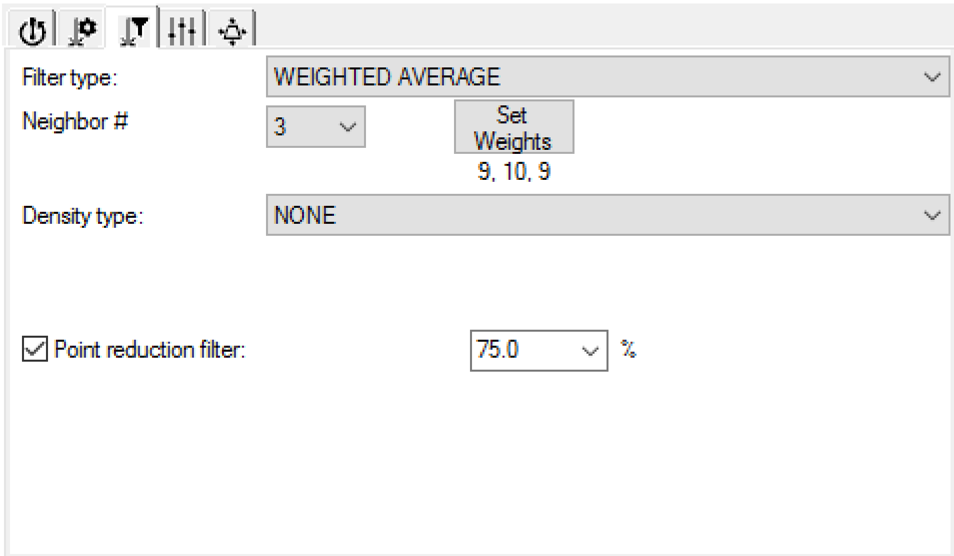


Example of Point Filtering of 50%



Filter Type: Weighted Average

NOTE: This is only available for Perceptron sensors.

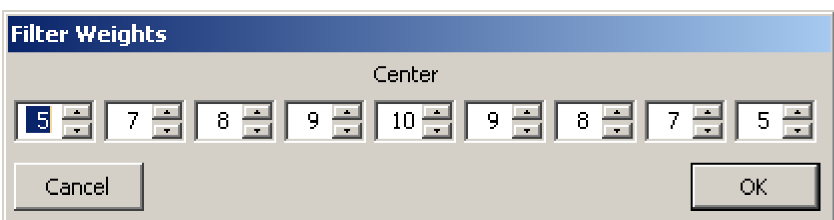


Weighted Average Filter Type

The **Weighted Average** filter smooths stripe data by computing a new location for each pixel. For each pixel in the stripe, this filter will use a weighted average of its neighboring pixels to compute a new location. This is the default filter.

Neighbor #: This value determines the number of total pixels considered when calculating a new location of any given pixel in a single stripe.

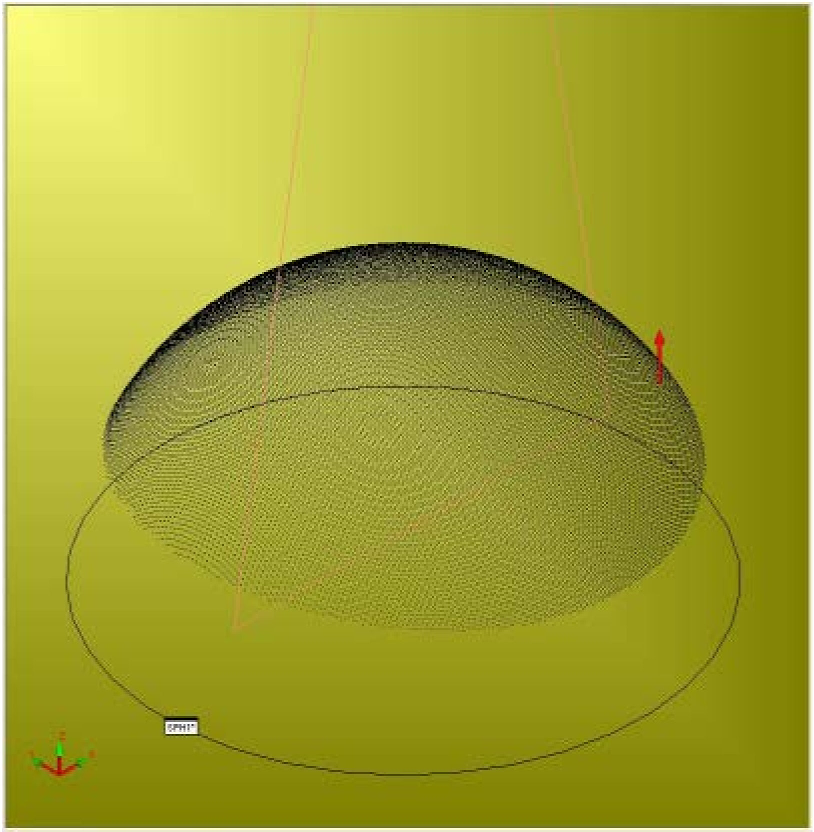
Set Weights: This button sets the relative importance of a given pixel's neighbor.



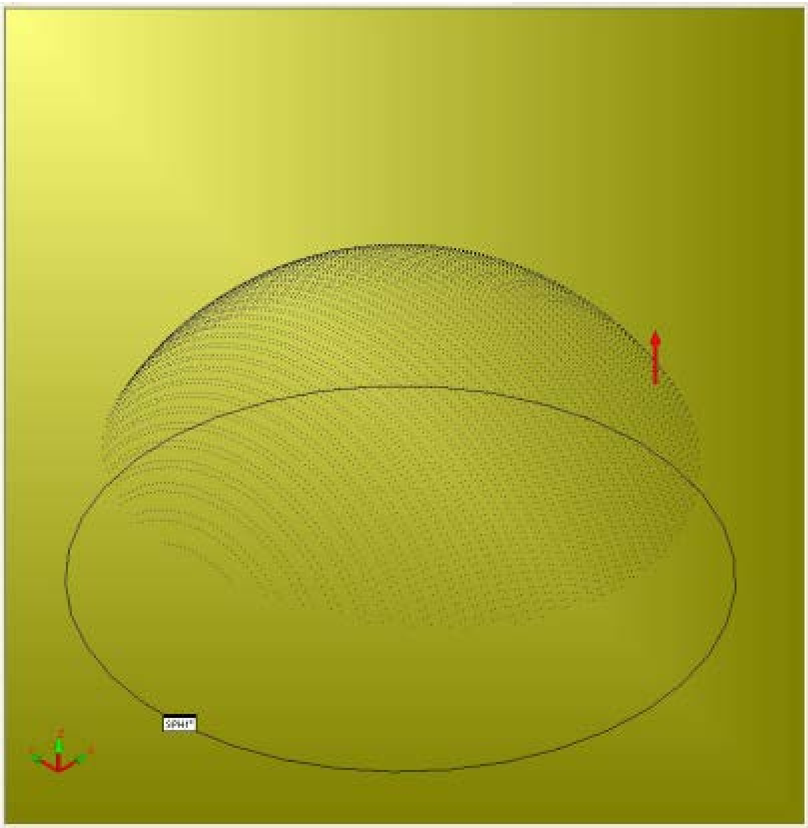
Use the up and down arrows for each of the pixel locations. Click **OK** to save your changes or **Cancel** to close without saving.

Point Reduction Filter: This check box determines whether or not PC-DMIS filters points along the scan line. If marked, you can select the desired percentage of total points to filter. If cleared, the complete data set is acquired without any filtering.

Example of Point Filtering Disabled



Example of Point Filtering of 50%

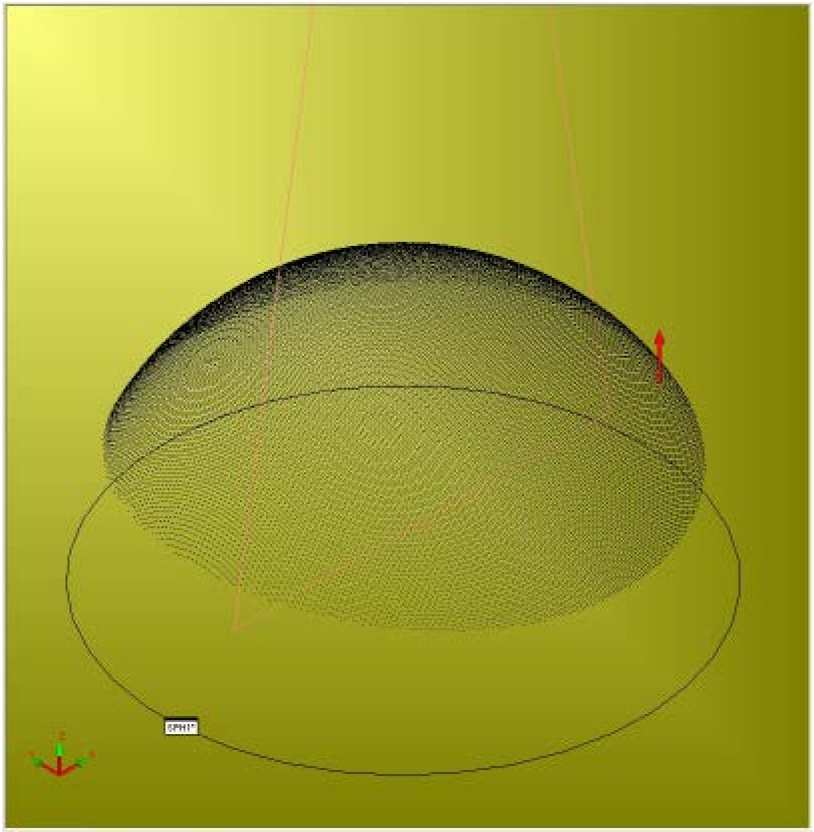


Filter Type: Stripe

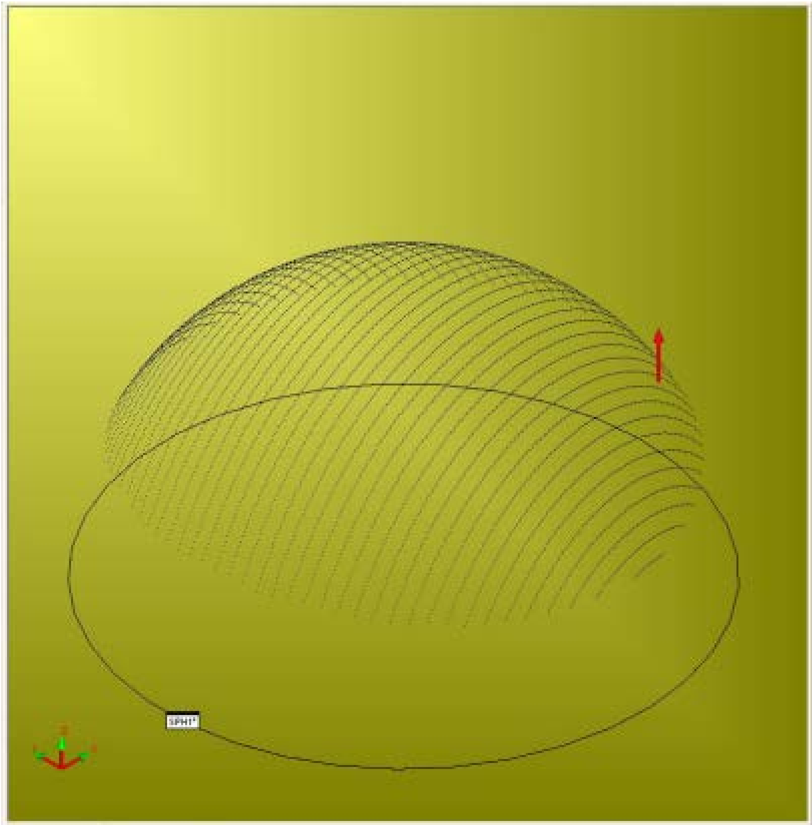
NOTE: This is only available for CMS sensors.

The **Stripe Filter** list lets you filter scan lines along the scanning direction. You can select a number from a scale of 1 to 10 (1 represents minimum filtering while 10 represents maximum filtering). If disabled, this acquires the complete data set without any filtering.

Example of Stripe Filtering Disabled



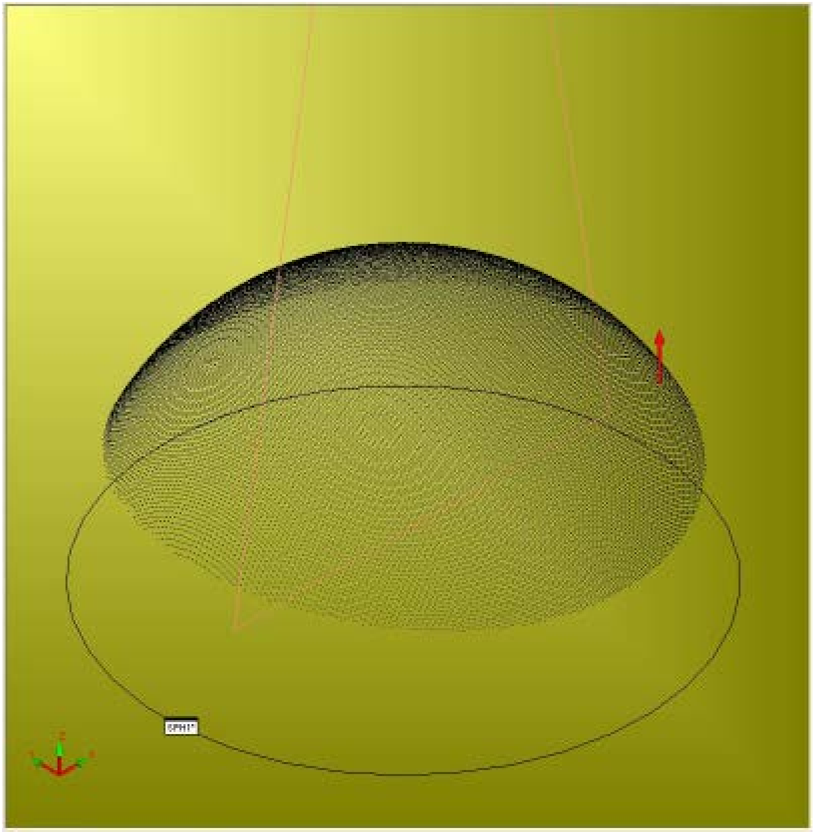
Example of Stripe Filtering of 5



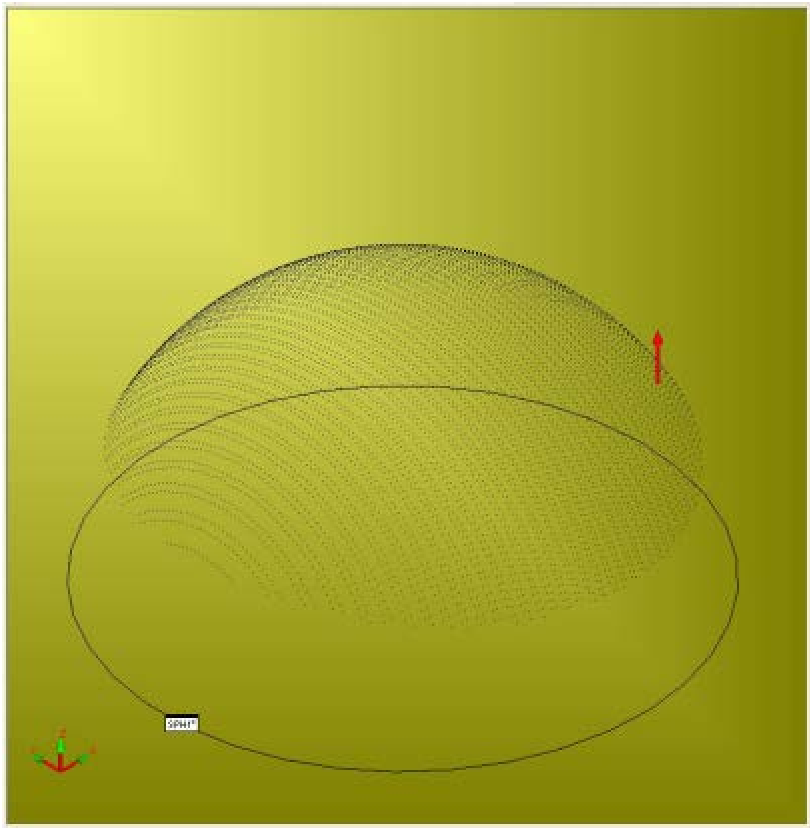
> **NOTE:** If you're using a CMS sensor with the Perceptron Toolkit as the Feature Extractor, the Auto Feature Square Slot feature only allows odd-numbered stripe filters (1,3,5,7,9).

Point Reduction Filter: This check box determines whether or not PC-DMIS filters points along the scan line. If marked, you can select the desired percentage of total points to filter. If cleared, the complete data set is acquired without any filtering.

Example of Point Filtering Disabled

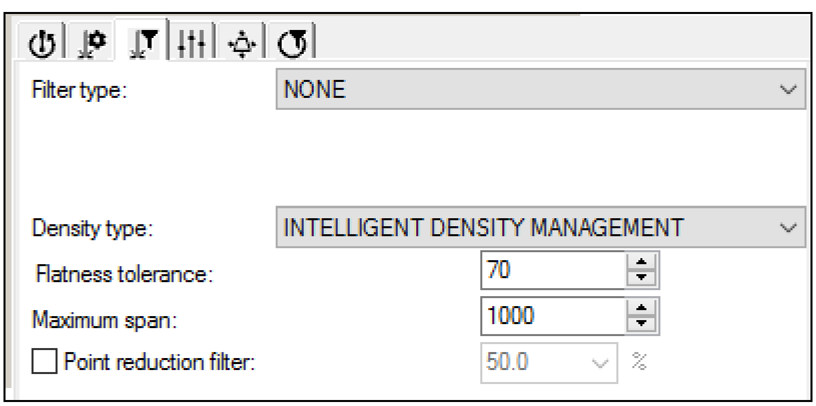


Example of Point Filtering of 50%



Density Type: Intelligent Density Management

NOTE: This is available only for the Perceptron Contour V5 sensor.



Intelligent Density Management with Filter Type - None

Intelligent Density Management (IDM) is *only* available for Perceptron V5 laser sensors. You can only scan at high speeds with IDM. You can use features scanned with IDM for auto feature extraction because edge points are found with IDM.

You can use **Filter Type** and **Density Type** together. For example, you might want a "Long Line" filter with IDM density. However, if you only want to apply the IDM density, set the **Filter Type** to **None**.

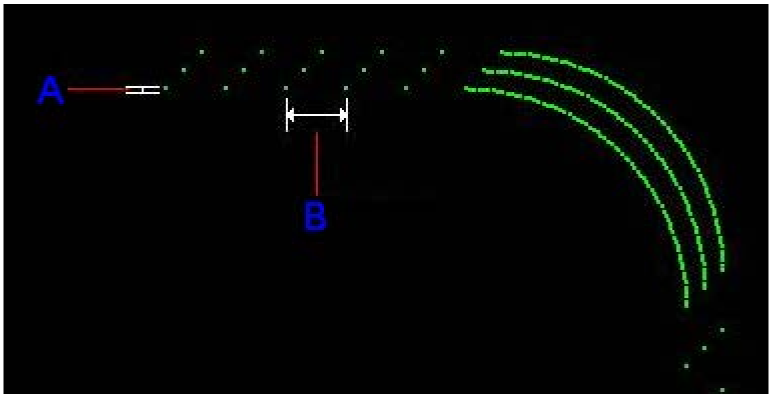
The two IDM settings work together to determine which points to reduce (remove) based on the position of the neighboring points. When data points are considered to be on the same plane, only a few points are needed. IDM retains points if they are outside the **Flatness Tol** or if the **Max Span** distance has been reached.

EXAMPLE: In the image below, you can see that IDM retains fewer points along the straight lines than along the curved lines.

IDM uses the following settings:

Flatness Tol (A): Provides a tolerance distance in microns. If neighboring points exceed this distance, IDM considers those points to not reside in the same plane. Points that deviate from this range are included in the subset of points. This value should be between 1-60.

Max Span (B): Provides the maximum distance (in microns) that included points can be from each other. Once the **Max Span** has been reached for points that are within the **Flatness Tol** a new point will be included in the subset of points. This value should be between 150-2500.



*IDM Sample - **Flatness Tol** (A) and **Max Span** (B)*

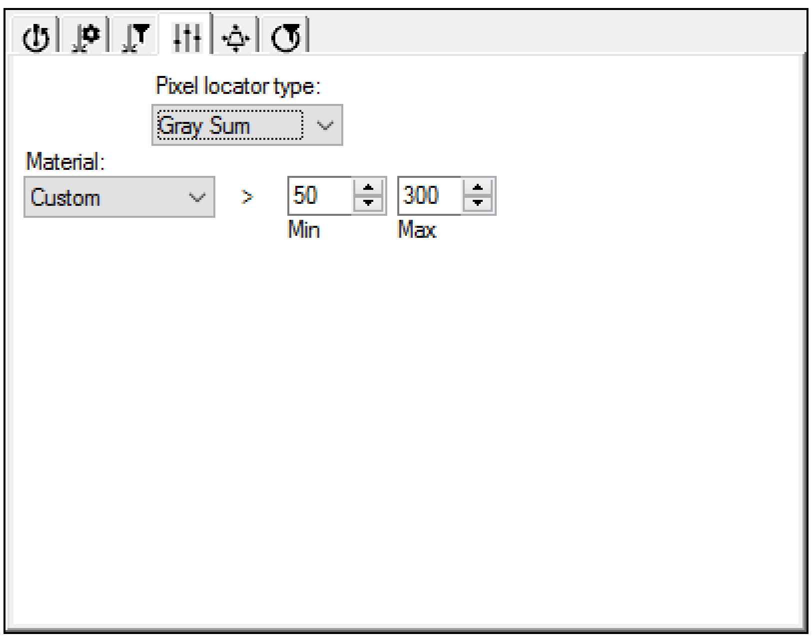
Example IDM Settings

Flatness Tol.	Max Span	Result
15	1000	Provides data at nominal 1mm point-spacing. This allows you to achieve significant data reduction without sacrificing surface details. This is the "optimal data compression"

		because it provides a good balance of CPU load, memory usage, and graphic card load.
150	2500	This is the maximum data-reducing IDM setting. This setting places a heavy load on the CPU, but it reduces memory usage and graphics card load.
1	60	Emulates V4 probe performance with a V5 probe. This setting is easy on the CPU, but require more memory and puts an increased load on the graphics card.
1	120	This essentially turns off IDM.

Laser Probe Toolbox: Laser Pixel CG Locator Properties tab

CAUTION: Only advanced users in specific situations should use the **Laser Pixel CG Locator Properties** tab.



Probe Toolbox - Laser Pixel Locator Properties tab

NOTE: Scanning methods with a portable device using a Perceptron laser differ from DCC machines. If you open the **Auto Feature** dialog box and are using a portable device with a Perceptron laser, the **Laser Pixel CG Locator Properties** tab is hidden.

The **Laser Pixel CG Locator Properties** tab only appears if you have a Perceptron laser sensor. This tab uses various mathematical algorithms to change how the software accurately determines the pixels comprising the stripe.

The algorithms operate on an image that consists of rows and columns of pixels. The laser stripe within that image illuminates a band of pixels. The pixel locator then computes the location of the true pixel in the image.

In the following pixel locator algorithms, PC-DMIS computes a surface point based on the illumination of a column of pixels in the image:

Gray Sum: If you select this locator type, PC-DMIS limits the data collection to the parts of the line that fall between the specified **Min** and **Max** values. These minimum and maximum limits are a percentage of the average intensity for each laser line. These limits can be used to improve the data quality for specific part geometry situations. See "Feature and Material Settings".

> **Material:** This list allows you to select a predefined material type (**Custom**, **Sheetmetal**, **White**, **Blue**, **Black** and **Aluminum**) with its corresponding Min/Max values. When you select a material type, the software loads the saved Min/Max values for that material type. Using the default option of **Custom** you can define a generic set of Min/Max values. If you modify the Min/Max values, the **Material** type automatically switches to Custom.
>
> **Min:** If any part of the laser line's intensity *falls below* this value, the software won't use that part. In situations where the *edges* are important, you can reduce this value so more of the edge data is preserved as the laser wraps around the edges. For a *shiny part* with internal corners that cause reflections and noise in the data, you can increase this value to eliminate the "noise" generated by internal reflections.
>
> **Max:** If any part of the laser line intensity *exceeds* this value, the software won't use that part. In some situations where a part has many contours that you cannot easily follow, the laser reflects strongly. This causes localized over-exposures. Reducing this value may help to ensure that the overexposed areas do not provide bad data.

> **NOTE:** The software always selects Gray Sum for portable devices using the Perceptron V5 laser sensor.

Fixed Threshold: If you select this locator type, PC-DMIS discards all the data below the threshold and it computes the actual pixel location as the center of gravity of the remaining pixels within the column.

Gradient: If you select this locator type, PC-DMIS computes the actual pixel location. It looks at a column of pixels and finds where the slope changes direction. For each direction change, PC-DMIS creates a pixel.

Exposure and Gray Sum Settings by Feature and Material

Based on the feature type and the part material type the Exposure value found on the **Laser Scan Properties** tab and the **Min** and **Max** Gray Sum values found on the **Laser Pixel CG Locator Properties** tab should be adjusted according to the following table:

Exposure and Gray Sum Settings				
Feature Based				
Feature	Material	Exposure	Min Grey Sum	Max Grey Sum
Sphere	Tungsten Calibration Sphere	120	10	300
	Ceramic	80	10	300
Gap/Flush	Sheet Metal	150	30	300
	White	100	30	300
	Blue	120	30	300
	Black	450	10	300
Circle	Sheet Metal	100	50	300
	White	100	50	300
	Blue	120	50	300
	Black	450	30	300
	Aluminum	80	50	300
Slot	Sheet Metal	100	50	300
	White	100	50	300
	Blue	120	50	300
	Black	450	30	300
	Aluminum	80	50	300

Edge Point	Sheet Metal	100	50	300
	White	100	50	300
	Blue	120	50	300
	Black	450	30	300
	Aluminum	80	50	300
Plane	Sheet Metal	100	30	300
	White	100	30	300
	Blue	120	30	300
	Black	450	10	300
	Aluminum	80	30	300
Surface Point	Sheet Metal	100	30	300
	White	100	30	300
	Blue	120	30	300
	Black	450	10	300
	Aluminum	80	30	300

Exposure and Gray Sum Settings

Exposure and Gray Sum Settings During Calibration

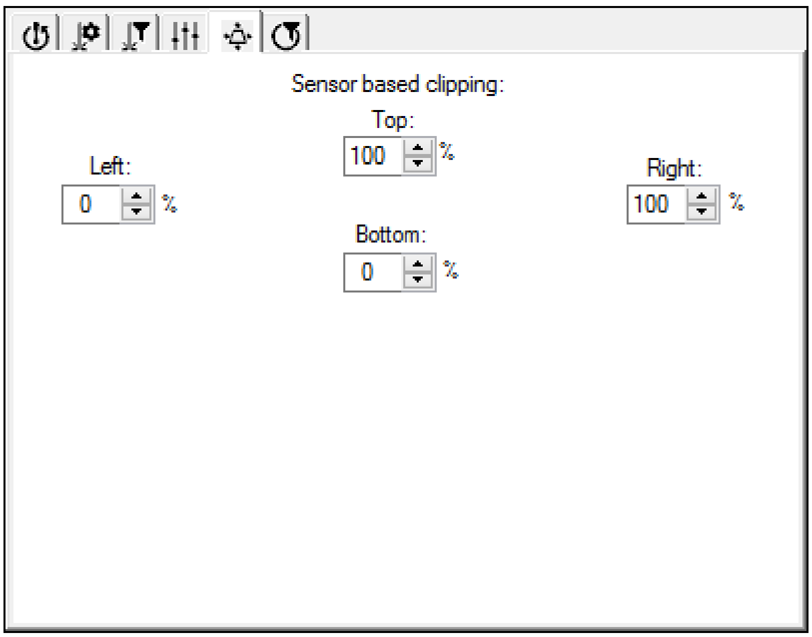
Prior to the start of calibration, PC-DMIS sets the exposure and gray sum values to the following:

- **Exposure:** 300
- **Gray Sum Min:** 10
- **Gray Sum Max:** 300

These are the settings that work best for most calibration scenarios. Once the calibration finishes, PC-DMIS restores your original exposure and gray sum values (from before calibration). While gray sum values of 10, 300 are often appropriate for calibration, values of 30, 300 are typical for normal scanning.

Also, the default exposure value of 300 is often not sufficient in rare lighting conditions (such using a V4i with sodium lighting). If PC-DMIS has difficulty accepting the arcs during the calibration process, you may need to raise the default calibration exposure value to 400 or so. In cases such as this, modify the `PerceptronDefaultCalibrationExposure` registry entry located in the **NCSensorSettings** section of the PC-DMIS Settings Editor. See the PC-DMIS Settings Editor documentation for more information.

Laser Probe Toolbox: Laser Clipping Region Properties tab



Laser Clipping Region Properties tab

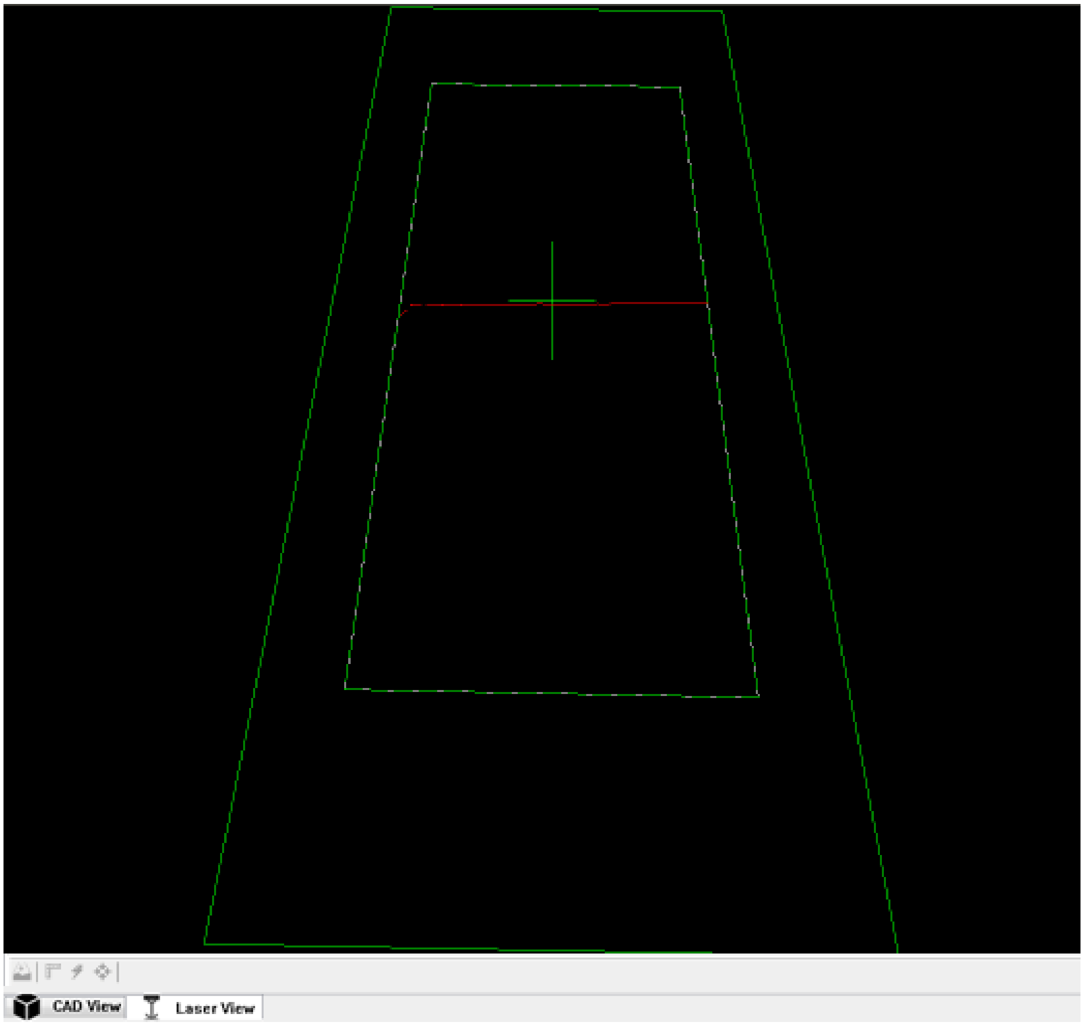
The **Laser Clipping Region Properties** tab allows you to set parameters to discard data outside a specified region, within the sensors field of view. This lets you keep only pertinent data.

Keystone: The large green trapezoid in the Laser View (see below) that represents the sensor's maximum field of view. The clipping region is within this field of view.

Sensor Based Clipping region: The smaller green trapezoid within the sensor's field of view.

The **Top**, **Left**, **Right**, and **Bottom** boxes can be set with values from 0 to 100 percent that allow control over the clipping region. This lets you discard data that is not needed.

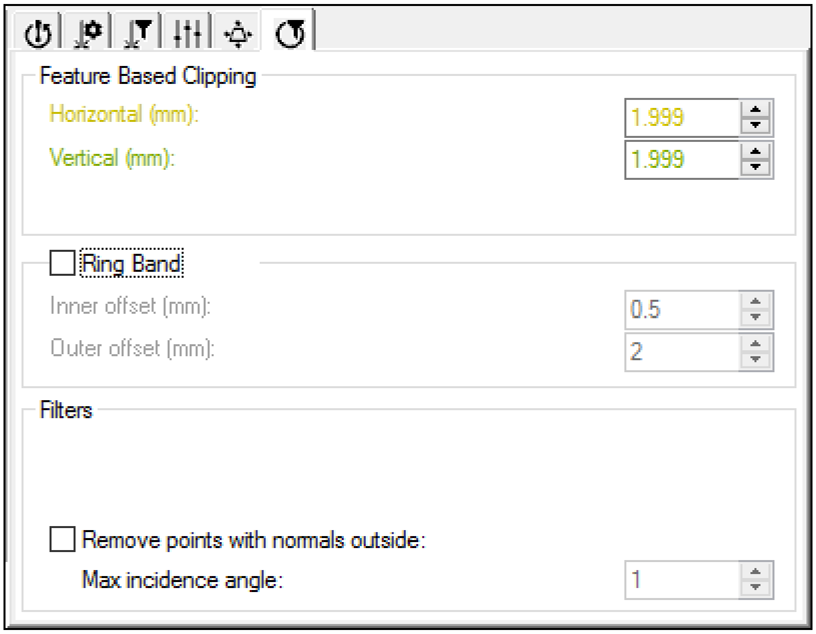
When the **Bottom** and **Left** values are at 0% and the **Top** and **Right** values are at 100%, the sensor will keep all of the data collected because the clipping region is the same as the maximum field of view.



Example of clipping data using Top 85, Bottom 85, Left 15, and Right 15

You can use the clipping region, for example, when measuring a hole. Since you wouldn't want data from a neighboring hole to interfere with the feature computation, you can control the area that is clipped, thereby discarding the unwanted data.

Laser Probe Toolbox: Feature Extraction tab



Feature Extraction tab

You can use the **Feature Extraction** tab to specify ring band and feature-based clipping parameters, as well as remove outliers on supported features.

NOTE: The **Feature Extraction** tab is only available when using a laser sensor.

Depending on the feature type, the following feature extraction parameters are available:

- Feature Based Clipping parameters: This is available for all auto features.
- Ring Band parameters: This is only available for Circle, Cone, Cylinder, Round Slot, and Square Slot, auto features.
- Filters:
    - Remove outliers parameter: This is only available for Surface Point, Plane, Cone, Cylinder, Sphere, and Flush and Gap auto features.
    - Remove points with normals outside parameter: This is only available for Surface Point, Plane, Circle, Round Slot, Square Slot, Polygon, Cylinder, Cone and Sphere auto features.

Also see "Extracting Auto Features from Pointclouds".

Feature Based Clipping Parameters

Feature Based Clipping area for non-plane auto features

PC-DMIS can clip laser data in both the horizontal and vertical directions when you type a distance in the **Horizontal** box and, when available, the **Vertical** box. This distance will clip all of the laser data outside of the defined distance, excluding that data when extracting the feature.

Alternately, for the Plane auto feature, you can clip data within an offset boundary around all of the CAD elements on a surface. This is also termed CAD segregation. See "CAD Offset" below.

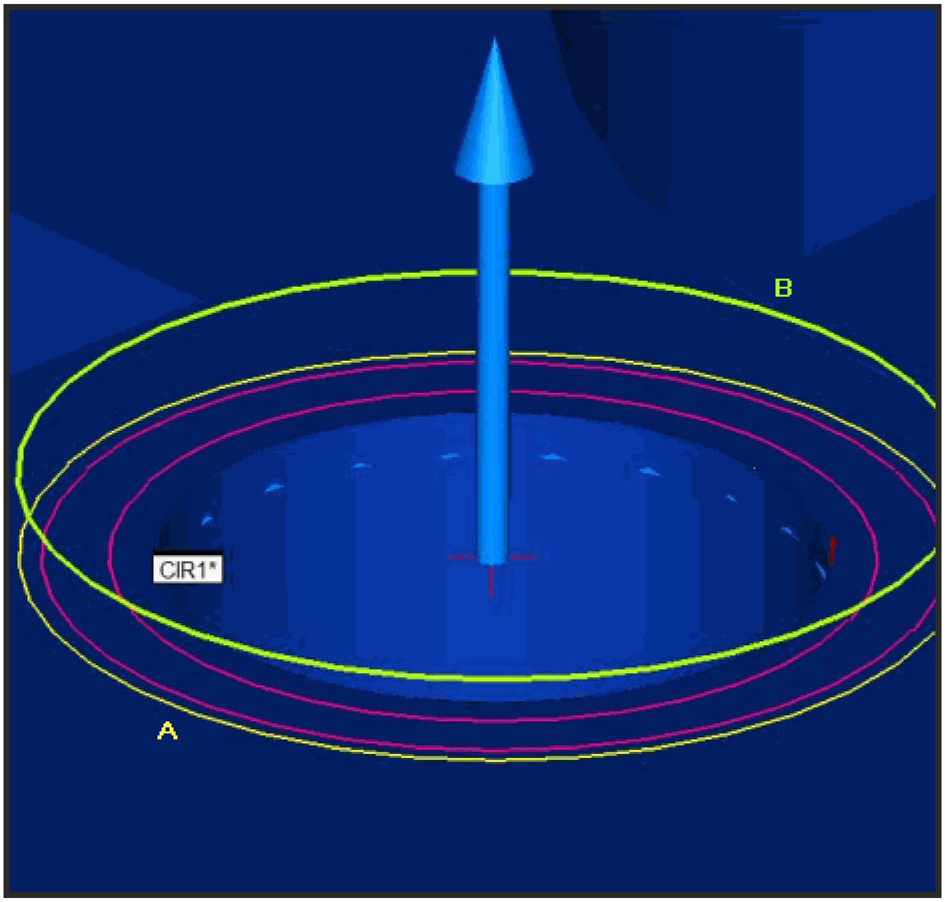
For the Cone auto feature, the value for Horizontal defines how much larger than the theoretical diameter is the circular boundary within which the feature points lie. The value for Vertical defines how much longer than the theoretical length is the cylindrical boundary within which the feature points lie.

Horizontal and Vertical Clipping

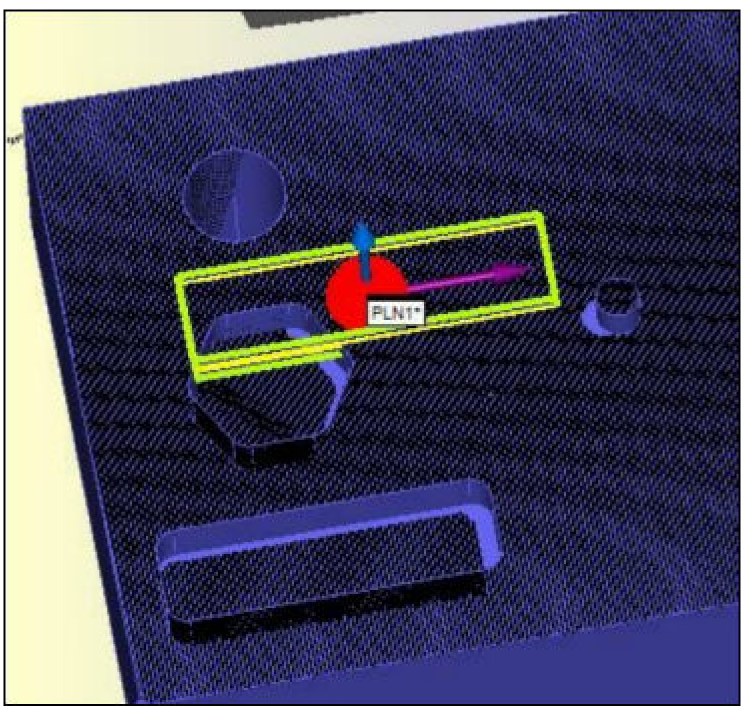
All of the auto features support horizontal clipping. These features support vertical clipping:

- Circle
- Cone
- Cylinder
- Polygon
- Edge Point
- Round Slot
- Square Slot
- Surface Point
- Plane

The clipping distances defined in the feature based clipping rings are shown as colored rings. The horizontal clipping appears as a yellow ring, and the vertical clipping appears as a light green ring.

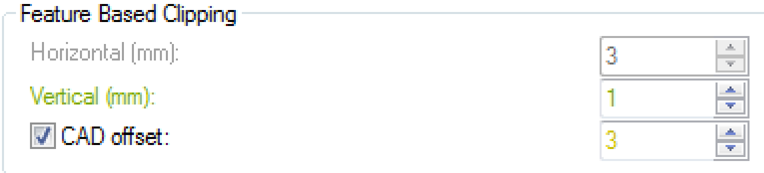


Sample Circle auto feature with horizontal clipping (A) and vertical clipping ring (B)



Sample Plane auto feature with horizontal and vertical clipping enabled

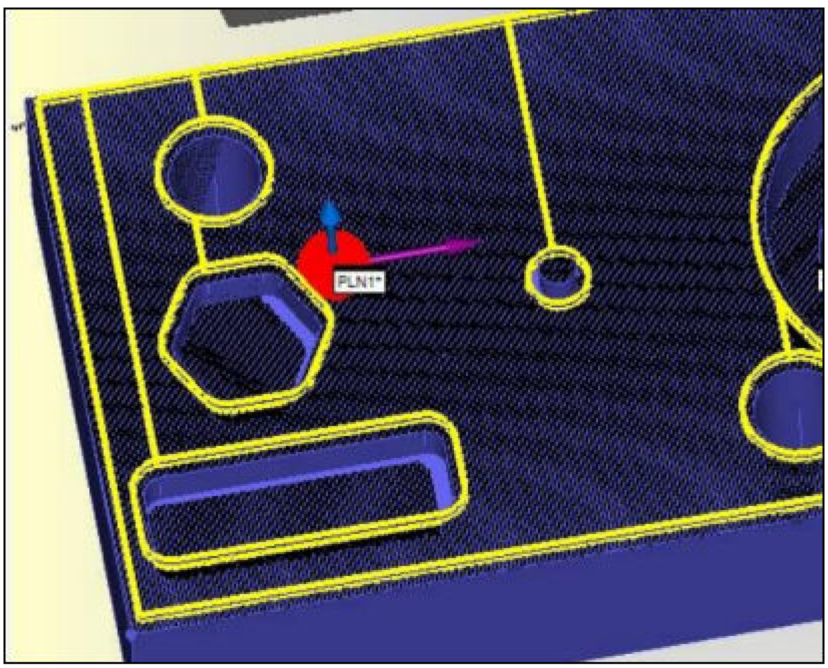
CAD Offset



Feature Based Clipping area for Plane auto feature

NOTE: The **CAD** check box and **Offset** box appear only when using the Plane auto feature.

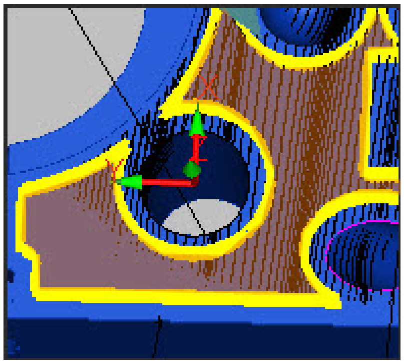
When you mark this check box, PC-DMIS creates a yellow offset boundary around each feature in the CAD model on the surface. The offset boundary is calculated by the **Offset** value. It is drawn at the specified distance away from the features and edges on the surface.



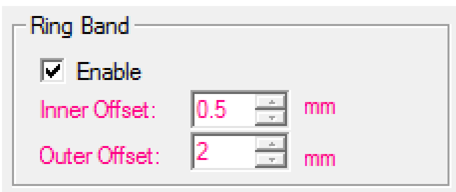
Sample Plane auto feature with CAD-based clipping enabled

PC-DMIS clips the laser data that falls inside of an offset boundary for all of the features in the CAD model on a surface. The data outside of the offset boundary is used to solve the plane.

For example, consider the image below, which shows a section of a sample part. The translucent orange overlay, added to the image here for clarification only, indicates the data PC-DMIS would use to create the Plane auto feature:



Ring Band Parameters



Feature Extraction - Ring Band

The **Ring Band** area is used to calculate the feature's projection plane and normal vector. The feature data is projected up into the ring band's plane. The following **Ring Band** controls are used to accomplish feature extraction for circles, round slots and square slots:

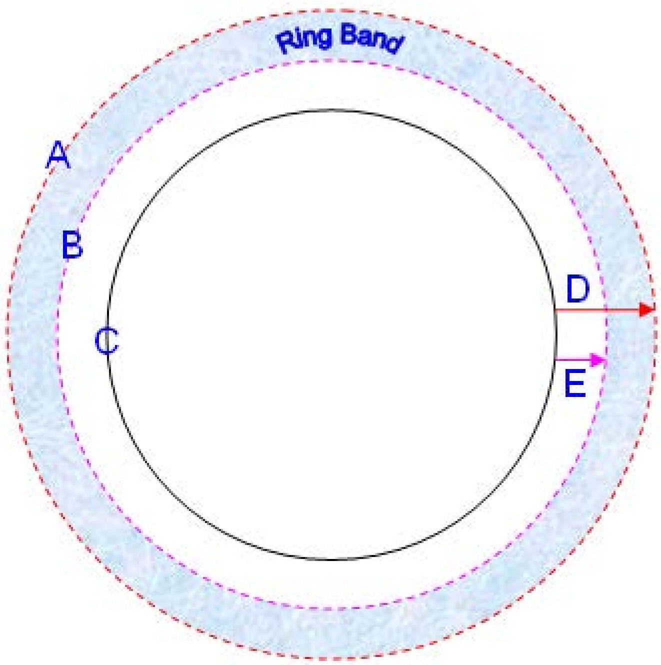
Enable - When this option is selected, the **Ring Band** options take effect.

The following default values are used when Auto Circle, Auto Round Slot, and Auto Square Slot are disabled:

- **Inner Offset** = 0.4x the theoretical diameter value
- **Outer Offset** = **Inner Offset** value + 3mm

Inner Offset - Provides the offset from the theoretical feature radius or form for the *inner* edge of the ring band. This value is expressed in measurement routine units and must be greater than or equal to zero (a value of zero means an inner edge of the ring band coincides with the feature nominal.) See Image below.

Outer Offset - Provides the offset from the theoretical feature radius or form for the *outer* edge of the ring band. This value is expressed in measurement routine units and must be greater than the **Inner Offset** value. See Image below.



(A) Ring band Outer Edge

(B) Ring Band Inner Edge

(C) Feature Theoretical value

(D) Outer Offset

(E) Inner Offset

Filters

Feature Extraction - Filters area

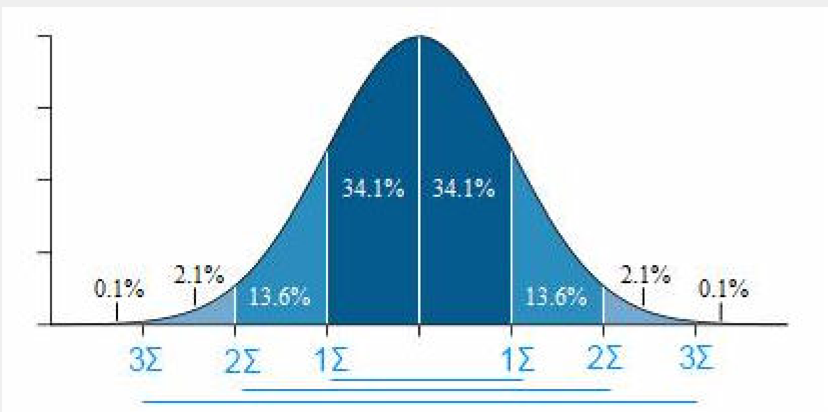
Remove outliers: When you select this check box, it excludes outliers from the feature based on the value for the **Standard deviation multiple** option. The **Remove outliers** check box applies only to the Auto Cone, Auto Surface Point, Auto Plane, Auto Cylinder, Auto Sphere, and Auto Flush and Gap features.

- The feature extractor evaluates the feature internally two or more times on the first attempt to get the standard deviation based on all points.

- In successive attempts, it re-evaluates the feature using only the points that are in the range of the outlier multiplied by the Σ. The sigma is the range, in the Gaussian distribution of the deviations, where the 68.2% of the best points used for fitting the feature lie.

Standard deviation multiple: The value for this option defines the selectivity of the filter. It can be a generic real number that is greater than 0. If **m** is the selected value, it means that all the scan points which deviate from the extracted cone is greater than **m** x **Actual standard deviation** (that is, the standard deviation of the measured points with respect to the calculated feature) are cut off from the calculation. Therefore, the lower the value of **m**, the more selective the filter.

In the first evaluation, the standard deviation is evaluated on all points. In a normal distribution, this could be represented as follows:



This means that the best points are in the interval from 0 to 1Σ. For example, if you wanted to get points only in that range, you would need to specify an outlier value from 0 to 1. Worse solutions would be obtained if you used higher outlier values.

Remove points with normals outside:

When enabled, this setting compares the estimated normal of each scanned point within the clipping zone to the feature theoretical normal (or CAD surface for 3D features).

NOTE: This parameter is only available for laser Circle, Cone, Cylinder, Edge Point, Flush and Gap, Plane, Polygon, Round Slot, Sphere, Square Slot and Surface Point auto features. The Edge Point and Flush and Gap features use the 2D filter method.

When measuring the laser feature, this filter is used to exclude scanned points which are on the opposite side of the part or on adjacent surfaces. The smaller the **Max incidence angle**, the more points are excluded.

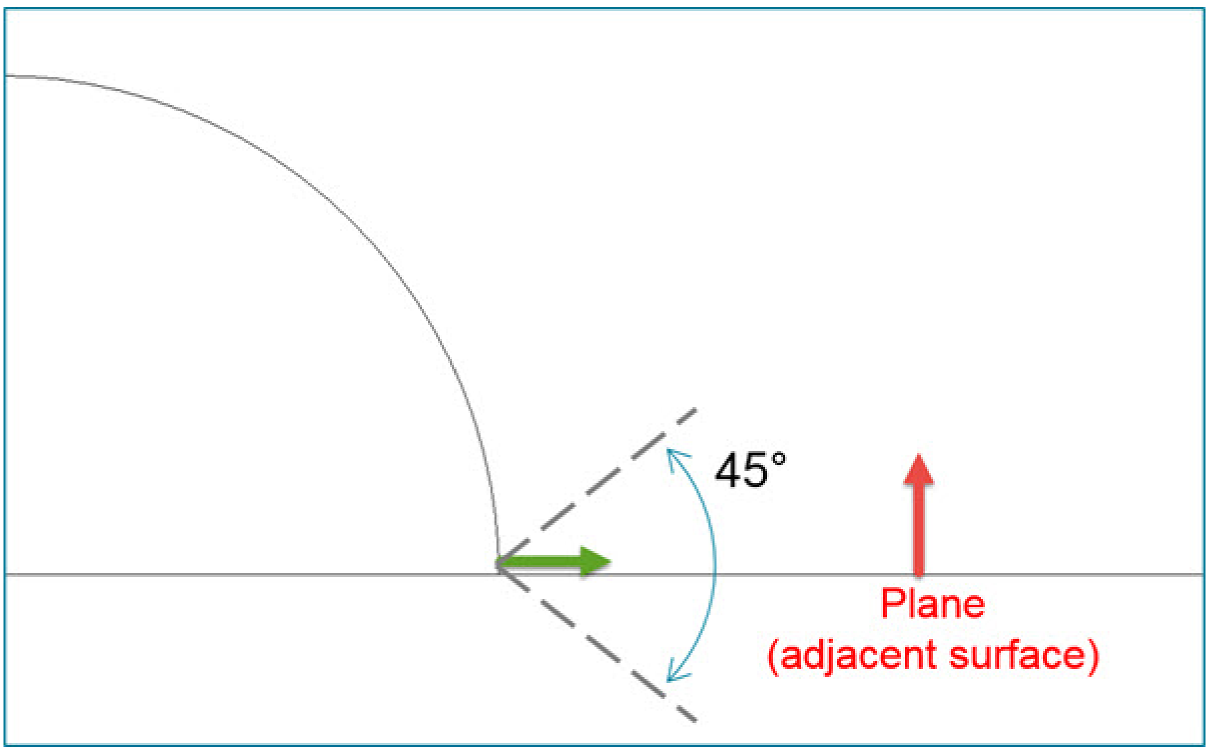
The effect of the **Max incidence angle** filter is enabled when the **Hide/Show segregated points** button is enabled from the Laser **Auto Feature** dialog.

3D Features Using the Max Incidence Angle

Laser Auto Features have a Horizontal and Vertical Clipping Zone. All scanned points within the clipping zone are initially evaluated.

For 3D features (Surface Point, Plane, Cylinder, Cone, and Sphere), this setting compares the estimated normal of each scanned point to the feature theoretical normal, or the vector of the CAD surface if a CAD model is used.

Points with a vector which fall outside this angle are excluded when measuring the feature.

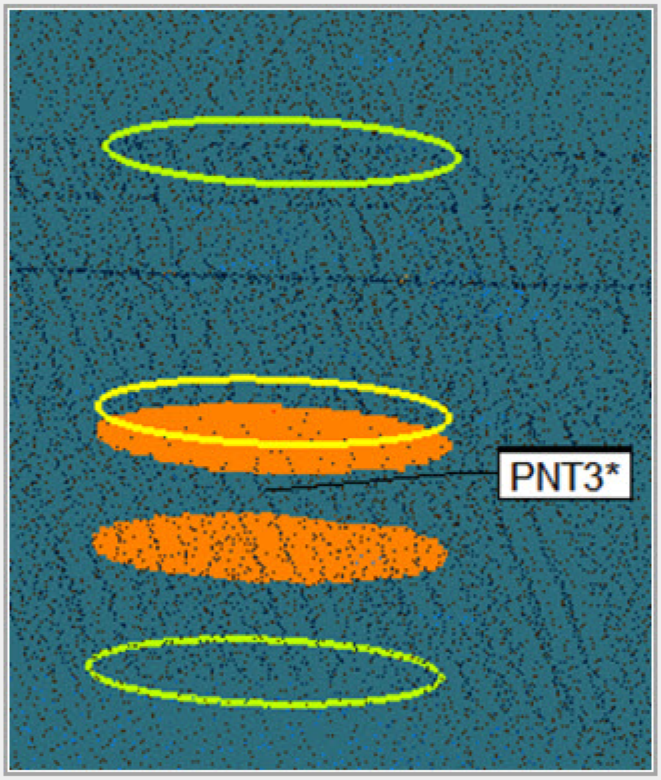


Example of the applied to a 3D Laser Auto Point feature

> **EXAMPLE:** On a thin sheetmetal part which was scanned from both sides, a Laser Auto Surface Point was created.

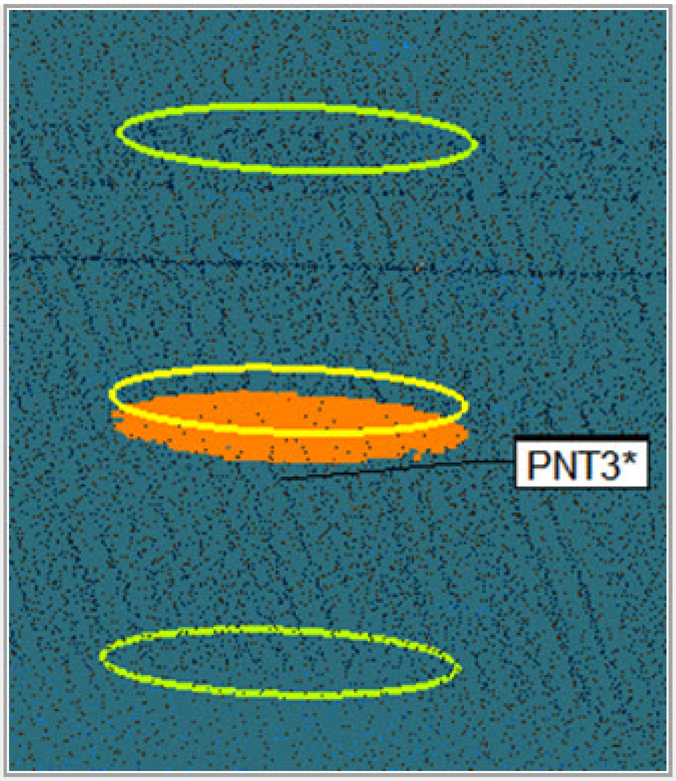
The Feature Extraction - Vertical clipping zone is set so it includes the part deviations, which in this case are larger than the sheetmetal thickness.

In this image, the scan uses no **Max incidence angle**:



Since the normals of the scanned points are not taken into consideration, the extracted point uses data from both sides of the part.

In this image, the scan uses a **Max incidence angle** of 60 degrees:

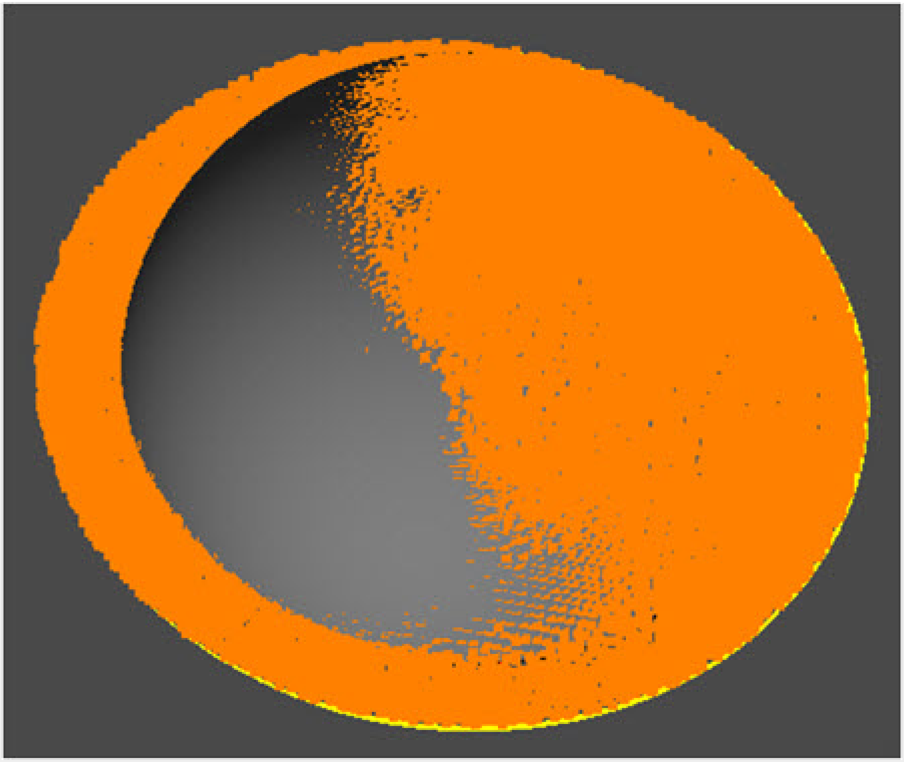


The software compares the estimated normal of each point in the clipping zone to the laser auto surface point theoretical normal. Points which fall outside this angle are not used for the feature calculation.

Example of the applied to a 3D Laser Auto Sphere feature

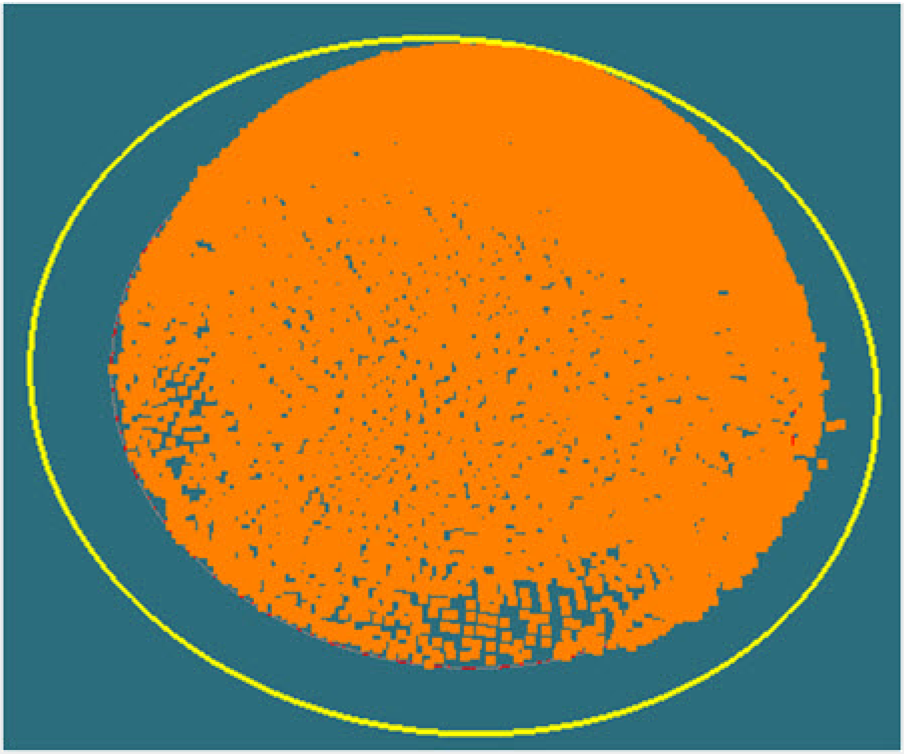
EXAMPLE: Laser extraction of a sphere previously required additional steps and manual selection to exclude adjacent surfaces.

In this image, no **Max incidence angle** is used:



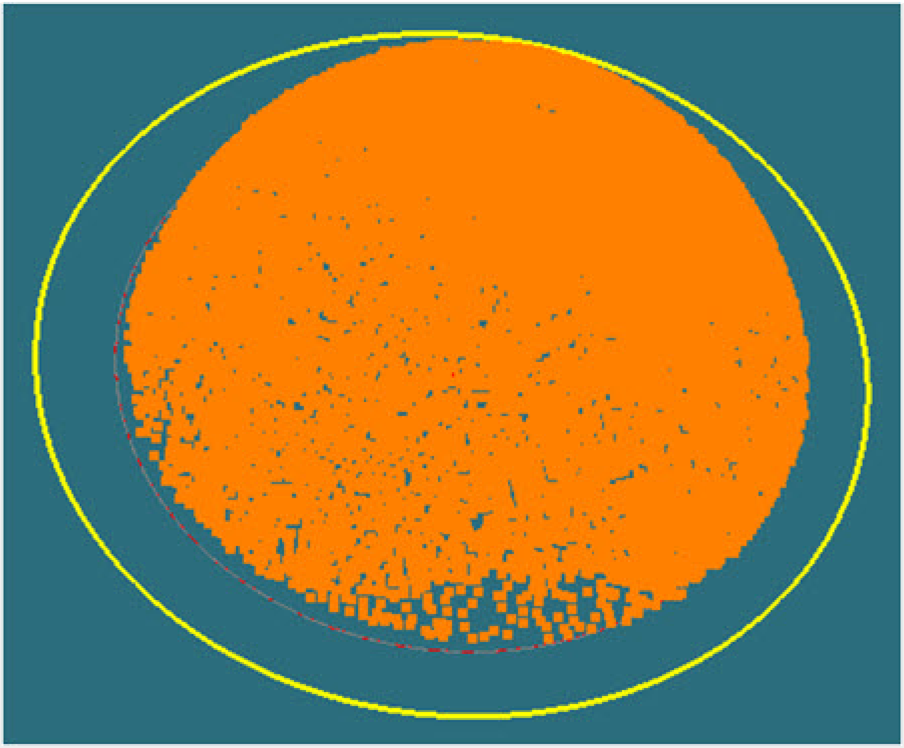
Data from the adjacent plane is used for sphere calculation.

In this image, a **Max incidence angle** of 60 is used:



A few outlying points are included.

In this image, a **Max incidence angle** of 45 degrees is used:

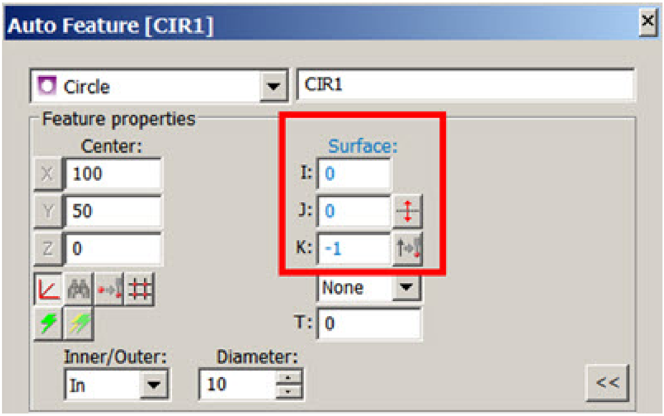


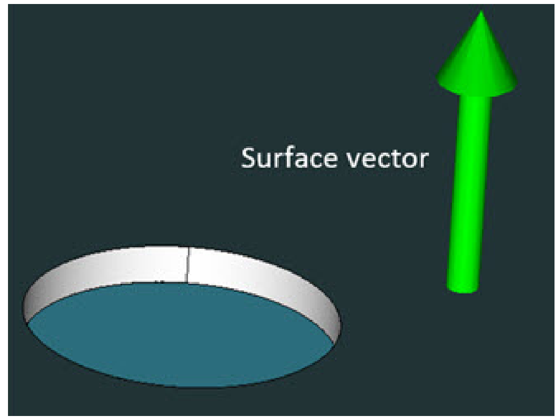
In this final example, the actual sphere data is best represented.

2D Features Using the Max Incidence Angle

Laser Auto Features have a Horizontal and Vertical Clipping Zone. All scanned points within the clipping zone are initially evaluated.

For 2D features (Circle and Slots), this setting compares the estimated normal of each scanned point to the feature theoretical Surface Normal.





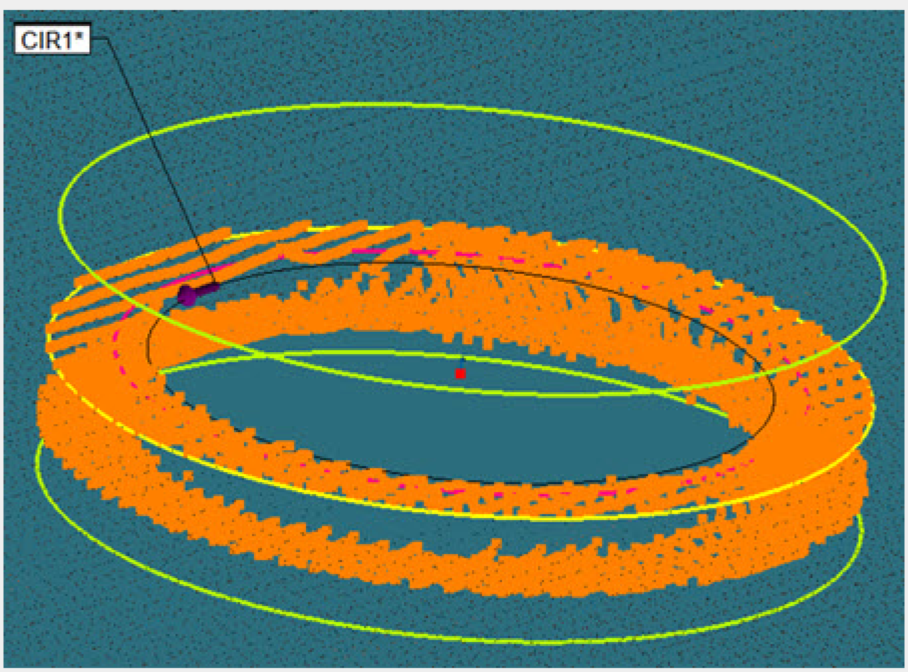
Points with a vector which fall outside this angle are excluded when measuring the feature.

Example of the Max incidence angle applied to a 2D Laser Auto Circle feature

EXAMPLE: On a sheetmetal part which was scanned from both sides, a Laser Auto Circle was created.

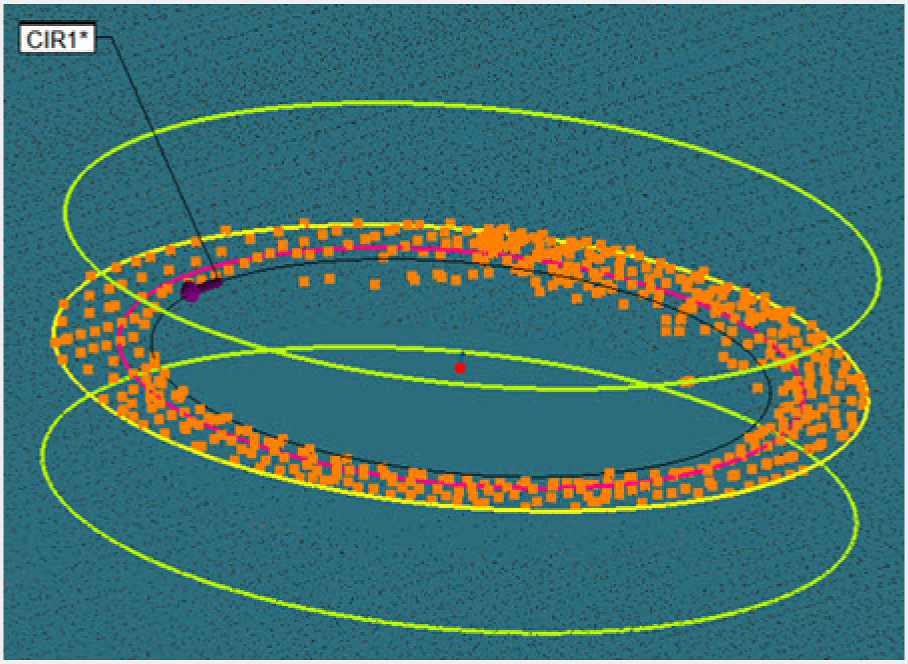
The Feature Extraction - Vertical clipping zone is set so it includes the part deviations, which in this case is larger than the sheetmetal thickness.

In this image, no **Max incidence angle** is used:



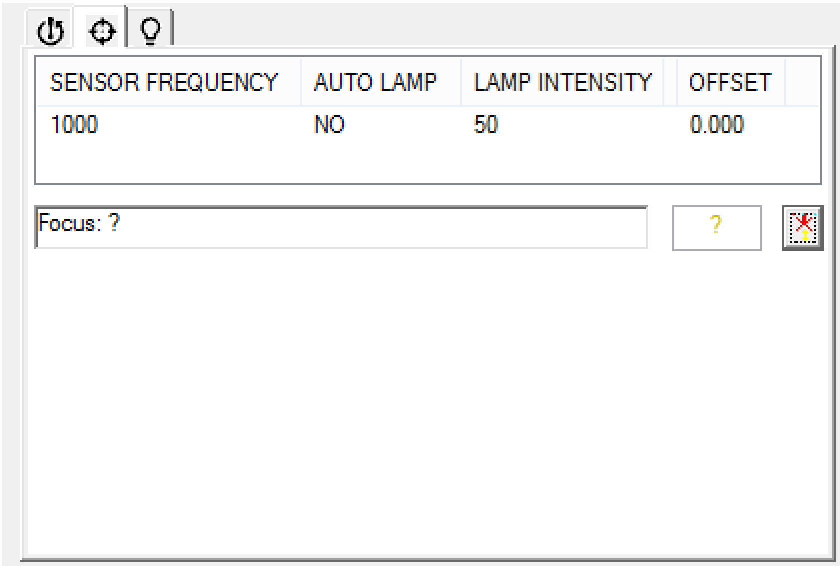
Since the normals of the scanned points are not taken into consideration, the extracted circle uses data from both sides of the part.

In this image, a **Max incidence angle** of 75 degrees is used:



The estimated normal of each point in the clipping zone is compared to the Laser Auto Circle, surface theoretical vector. Points with a vector which fall outside this angle are not used for the feature calculation.

CWS Parameter Probe Toolbox Dialog



CWS Parameter Probe Toolbox dialog

The CWS Parameter Probe Toolbox dialog is available once the system has been appropriately configured as described here:

- The CWS has to be configured as the active laser system. Usually this will be done locally by the factory during the startup procedure or by a Service Engineer.
- Once the system is correctly configured, you must define a probe with the correct properties. The probe is constructed using the **Probe Utilities** dialog box. You should use the OPTIVE_FIXED selection and a lens that includes CWS. This should be defined in the USRPROBE.DAT file. This is also usually provided locally by the factory.

The columns in the tab can include the following information:

+ TOLERANCE

Defines the upper Tolerance value for the measurement.

- TOLERANCE

Defines the lower Tolerance value for the measurement.

SENSOR FREQUENCY (Measurement Rate)

The measurement rate sets the number of measured values the optical sensor records per unit of time. For example, when the measurement rate is set at 2000 Hz, 2000 measurement values are taken per second. The intensity indicator on the display can help in selecting the correct setting.

Setting Range

As a rule, the user should strive to measure at the highest possible measurement rate in order to acquire as many measurement values in as little time as possible. In the case of surfaces with very low reflectivity, it may be necessary to reduce the measurement rate. This has the effect of illuminating the optical sensor‘s CCD-line longer and thus making it possible to perform measurements even if the reflected intensity is very low.

Overmodulation of the CCD-line on highly reflective surfaces and at small measurement rates can lead to measurement errors. If the intensity indicator displays a blinking **„Int: 999"**, overmodulation is occurring. When overmodulation occurs, the next-highest measurement rate should be selected. If the maximum measurement rate (2000 Hz on CHRocodileS, 1000Hz on CHR150E) is already set, the reflected intensity can be reduced in one of two ways:

- By positioning the sensing head in the upper or lower threshold of the measurement range
- By engaging the **autoadaptfunction** (where the **AUTO LAMP** parameter is set to **YES**). This will adapt the lamp intensity continuously depending on the part reflection. Here, a dark reference is not used. This is the method supported in PC-DMIS.

AUTO LAMP (Adjust Lamp Intensity)

Under adjust lamp intensity, the relative pulse duration of the LED and with it the effective brightness of the light source can be selected.

If, for example, a highly reflective surface is being measured, on which the highest measurement rate still results in overmodulation, then it makes sense to reduce the exposure time.

If a poorly reflecting surface is to be measured with a high measurement rate, this can be achieved by means of a longer pulse duration.

AUTO LAMP: NO

When the function is turned off, the current light intensity of the LED will be used.

AUTO LAMP: YES

The independent adjustment of flash time for the LED during an exposure time makes it easier for the user to automatically receive the best intensity settings when measuring on variable surfaces and with it an optimal signal-to-noise-ratio.

The brightness of the lamp is modulated such that a defined percentage of the modulation amplitude is achieved. The value can lie in the range of 0% to 75%. For most applications, a brightness value between 20% and 40% is recommended.

EXPOSURE TIME (Brightness Value)

If the **AUTO LAMP** parameter is set to **YES**, the exposure time (brightness value) can be selected here.

The brightness of the lamp is modulated such that a defined percentage of the modulation amplitude is achieved. The value can lie in the range of 0% to 75%. For most applications, a brightness value between 20% and 40% is recommended.

FILTER [SENSOR INTENSITY] (Detect Threshold)

Under **Set Detect Threshold**, the value for the threshold between noise and the measurement signal can be set. Peaks falling beneath this threshold are recognized as invalid and shown on the display as the measurement value “0”.

For a valid measurement, the intensity should fall between 0 and 999 on CHRocodileS or 99 on CHR150E; otherwise, the measurement rate must be changed.

If the distance to a surface with low reflectivity is measured, the intensity of the reflected light can be too low and the measurement rate must be reduced. For a measurement rate under 1 kHz, a threshold of 40 on CHRocodileS or 25 on CHR150E is recommended. This prevents measurement values of too low an intensity, which rise only slightly above the noise, which would falsify the measurement.

At a measurement rate of 1 kHz and higher (only for CHRocodileS), a threshold of 15 is expedient in fully exploiting the device’s dynamics.

OFFSET

This is the offset the machine will move in the measurement direction in addition to the measurement position.

Execution Modes

With PC-DMIS laser, you can use one of the following execution modes:

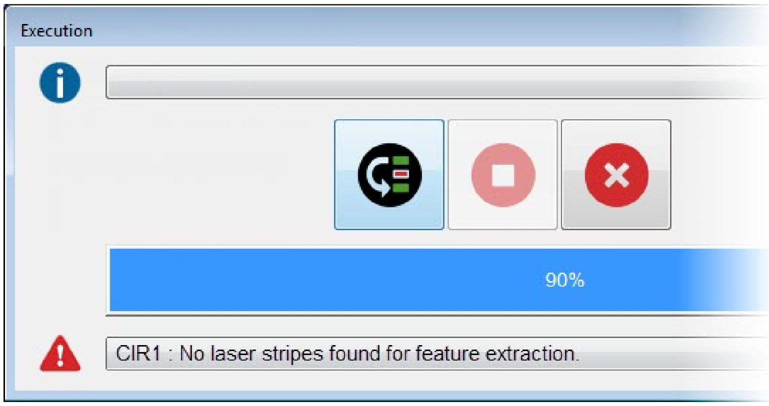
- Asynchronous Execution Mode (default mode)
- Sequential Execution Mode

Using Asynchronous Execution Mode

This is the default execution mode. In this mode, in order to speed up execution, the software ignores any feature calculation errors and proceeds to the next feature. If an error occurs during the execution of the measurement routine, the **Execution** dialog box presents you with these two options:

**Cancel** - This cancels the execution of the measurement routine.

**Skip** - This resumes the execution of the measurement routine from the next feature. The skipped feature command turns red in the Edit window.



Execution dialog box

Asynchronous Execution Mode Example

Suppose you have three circles in sequence in your measurement routine. This execution mode behaves as follows:

Scan CIR1.

Begin extraction of CIR1 from its pointcloud.

Scan CIR2.

Begin extraction of CIR2 from its pointcloud.

Scan CIR3.

Begin extraction of CIR3 from its pointcloud.

If CIR2 fails to extract, it generates its error, but because the default execution mode continues execution, the calculation error may appear in the **Execution** dialog box while the machine is already scanning CIR3 or maybe even a later feature. Use Sequential Execution Mode if you want to pause the execution when measurement errors occur.

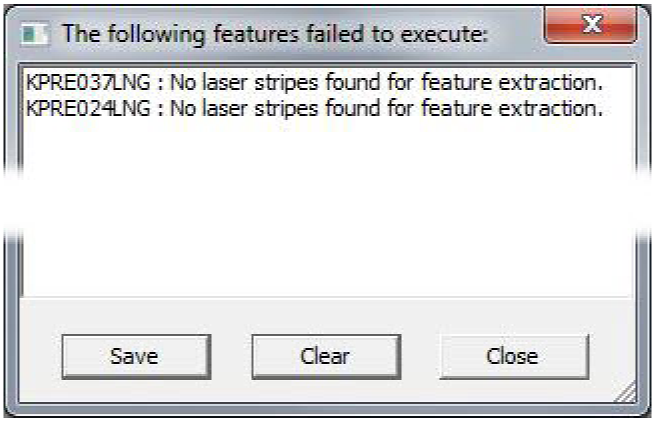
Using ONERROR with this Mode

In Asynchronous Execution Mode, if PC-DMIS encounters an error, and an ONERROR command has the SKIP parameter defined as shown below, it hides the **Execution** dialog box and skips the feature that had the error:

```
ONERROR/LASER_ERROR, SKIP
```

Unless there are critical errors, the SKIP parameter lets the measurement routine execute all the way through without anyone tending it.

After the entire measurement routine finishes execution, PC-DMIS displays the features that failed to execute in a dialog box. From that dialog box, you can click on any listed feature to locate the feature command in the Edit window and edit it as needed.



List of failed executed features dialog box

For detailed information on the ONERROR command, see the "Handling Laser Sensor Errors Using ONERROR" topic.

Using Sequential Execution Mode

In Sequential Execution mode, when the measurement routine measures and calculates a feature, it does not proceed with the execution until it finishes calculating the current feature. This execution mode allows you to have concrete information about the problem feature when an error message does appear. In addition, execution stops when a message appears. This may help avoid collisions with the part. Sequential execution is slower than the default mode (asynchronous execution), but it allows you to monitor errors as they occur.

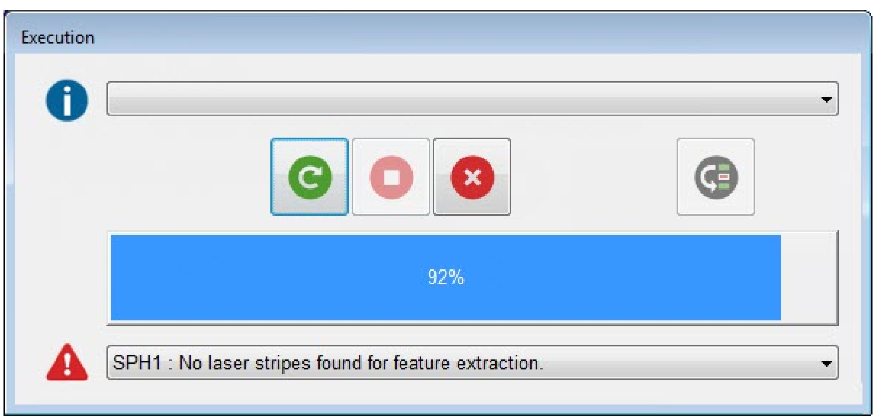
Generally, you should use this mode when executing a measurement routine for the first time, or when you want to test the machine's movements, laser parameters, or feature calculations.

If an error occurs during Sequential Execution, you have the following options in the **Execution** dialog box:

**Cancel** - This cancels the execution of the measurement routine.

**Skip** - This resumes the execution of the measurement routine from the next feature. The skipped feature command turns red in the Edit window.

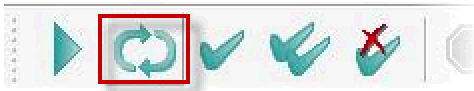
**Try Again** - This retries the execution beginning at the failed feature.



Execution dialog box

Enabling Sequential Execution Mode

To enable Sequential Execution mode, select **File | Execute | Sequential Execution** or from the **Edit window** toolbar, click the **Sequential Execution** icon.



Sequential Execution icon on the Edit Window toolbar

The software shows this icon in a pressed state when in Sequential Execution mode. PC-DMIS only stays in Sequential Execution for the current execution. Afterward, it reverts to the default execution mode.

About ONERROR Commands

ONERROR commands do not work with the Sequential Execution mode. PC-DMIS ignores any ONERROR command it encounters. For detailed information on the ONERROR command, see the "Handling Laser Sensor Errors using ONERROR" topic.

Using Sound Events

Sound Events provide audible feedback in addition to the visual user interface. This allows you to perform measurement actions if away from the screen. To access the **Sound Events** tab of the **Setup Options** dialog box, select the **Edit | Preferences | Setup** menu item.

When you work with a laser device, these Sound Event options are particularly useful:

Laser Manual Calibration Bottom - This sound plays when calibration measurements for a given field should be taken in the upper region of the sphere.

Laser Manual Calibration Field Counter - This sound plays to indicate in which field measurements should be taken during calibration.

- 1 Beep - Far
- 2 Beeps - Left
- 3 Beeps - Right

Laser Manual Calibration Top - This sound plays when you need to take calibration measurements for a given field in the lower region of the sphere.

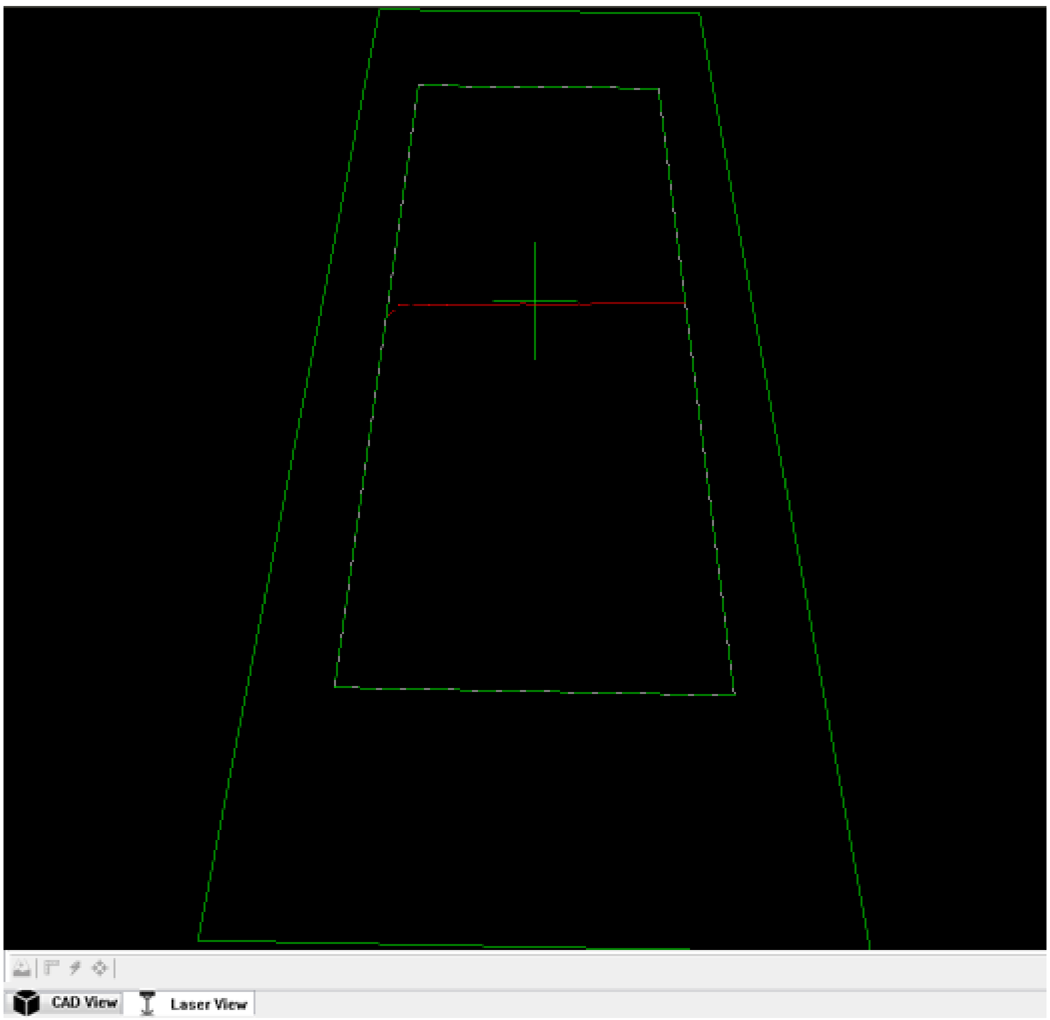
Laser Sensor Initialization End - This sound plays at the end of the laser sensor initialization.

Laser Sensor Initialization Start - This sound plays at the beginning of the laser sensor initialization.

Laser Scan - This sound plays for each new step of the sensor calibration.

Using the Laser View

You will use the **Laser View** tab during the laser probe calibration, scanning, and auto feature measurement. The **Laser View** tab of the Graphic Display window helps you visualize what the sensor "sees". It shows what information will be used. Remember, any data outside the clipping region rectangle will be disregarded during the scanning process. See the screen capture in "Laser Probe Toolbox: Laser Clipping Region Properties tab" for more information.



Graphic Display window - Laser View tab

Click the **Start/Stop** button  to toggle the On or Off state of the laser as viewed from the **Laser View**. When any changes have been made in the **Probe Toolbox**, the laser state needs to be toggled for the changes to be applied in the **Laser View**.

Perceptron Sensor Additions:

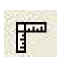
Toggle AutoExposure - Clicking this button while the laser is aimed at the part, PC-DMIS automatically determines the optimum exposure to be used for measurement. See "Exposure".

Perceptron and CMS Sensor Additions:

If you are using a CMS or Perceptron sensor, these buttons appear:

AutoClip - Automatically sets the clipping according to the data present in the Laser View tab.

Reset Clipping - Erases existing clipping, returning the entire sensor view for the selected scan zoom mode. See "Scan Zoom Modes (for CMS Sensors)".



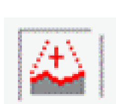
Ruler - Centers the part in the sensor's field of view.

In addition, for Perceptron and CMS sensors, you can drag the clipping region with the mouse. This provides an easy-to-use alternative to adjusting the clipping region by typing values in the **Probe Toolbox**.

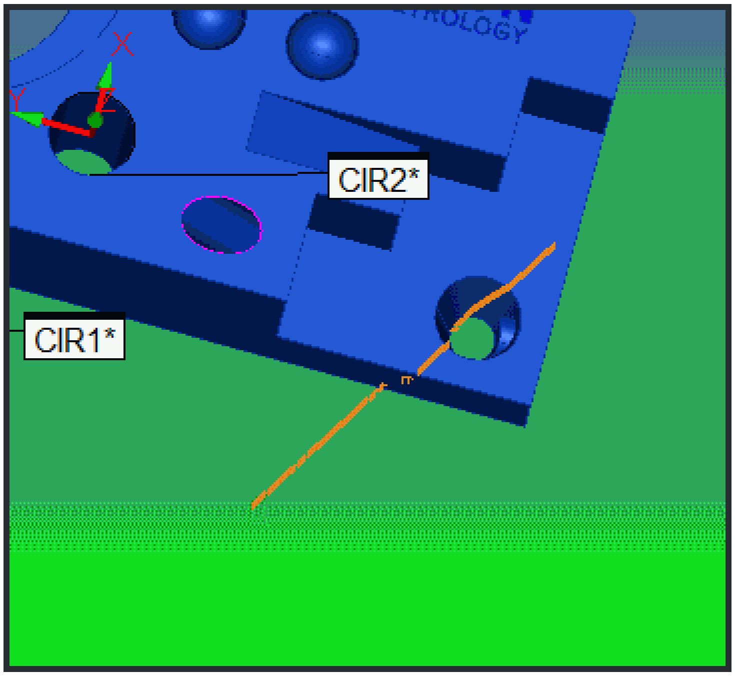
Using the Scan Line Indicator

PC-DMIS Laser displays a colored scan line indicator in the Graphic Display window to represent the location of the actual beam's scan line in 3D space. The indicator only functions when you run PC-DMIS in on-line mode with an actual laser sensor that points at the part in real time.

Click the **Start / Stop Laser View** icon in the **Laser View** tab to turn the scan line indicator (and the Laser View) on or off.

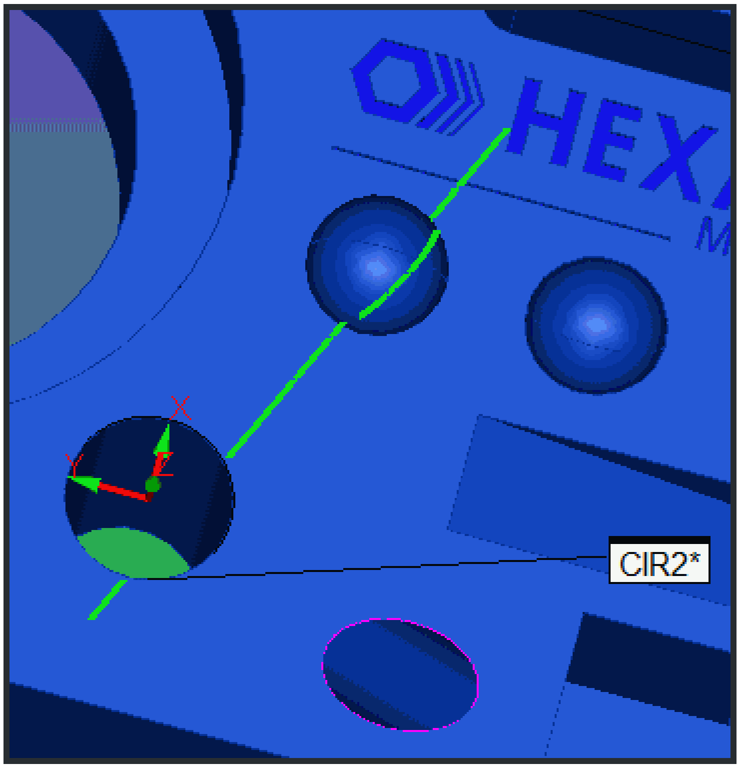


If the beam is within range, it appears in the Graphic Display window and blinks whenever the laser beam pulses. As the beam move towards the part, the indicator begins to change colors. As it nears the desired focal range, it changes colors from red, to orange, then yellow, then yellow-green, and finally to green.



A sample Scan Line Indicator (in orange) shows that the beam's scan line position is too far above the part.

This green color signifies that the beam is at the optimal distance away from the part for scanning.



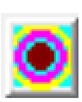
A sample Scan Line Indicator (in green) shows that the beam's scan line position is at the optimal focal distance

If you move the beam too close to the part, it again moves away from the desired green color toward a red color.

Understanding the Visualization Tools

PC-DMIS provides you with graphical overlays that it draws on top or around features you create or edit in the Graphic Display window. These colored overlays give a visual perspective for matching colored parameters or settings in the **Probe Toolbox** and in the **Auto Feature** dialog box.

You can turn these visualization overlays on or off with the **Visualization Tools ON/OFF** icon from the **Laser Scan Properties** tab of the **Probe Toolbox** (**View | Other Windows | Probe Toolbox**).



Visualization Tools ON/OFF icon

Below are some examples. These cover all the possible graphical overlays.

Explanation of Colored Overlays

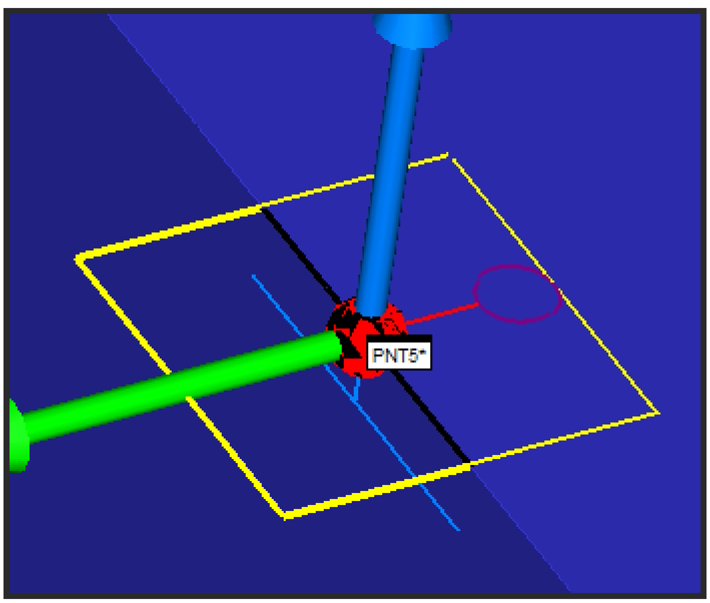
- **Yellow Line or Circle** - The **Overscan** region.
- **Blue Line or Circle** - The feature's Depth value.
- **Red Line** - The feature's Indent value.
- **Purple Circle** - The feature's Spacer value.
- **Pink Circles or Pink Rectangles** - The feature's **Ring Band** value.

Cones and Cylinders Overlays

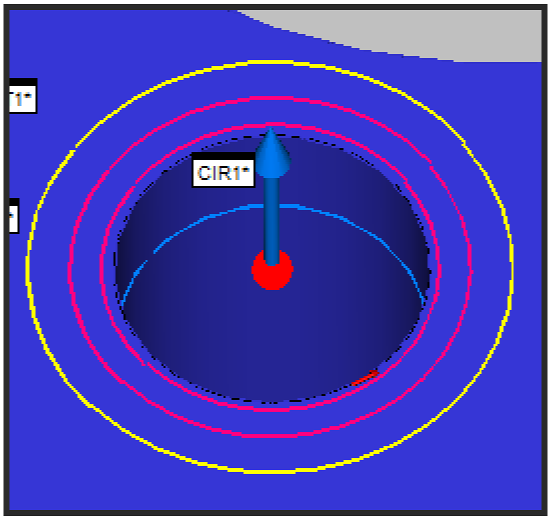
- *DCC Cylinders and Cones* show their bounds (the start and end points plus the **Overscan** value) in a light sea-green color. See the image of the sample DCC cone below.
- *Portable Cylinders and Cones (or Feature Extraction only features)* show their bounds (the start and end points minus the **Vertical Clipping** value) in a lime-green color. See the image of the sample portable cylinder below.

For information on specific parameters or features, see the appropriate topics within the "Creating Auto Features with a Laser Sensor" section of this documentation.

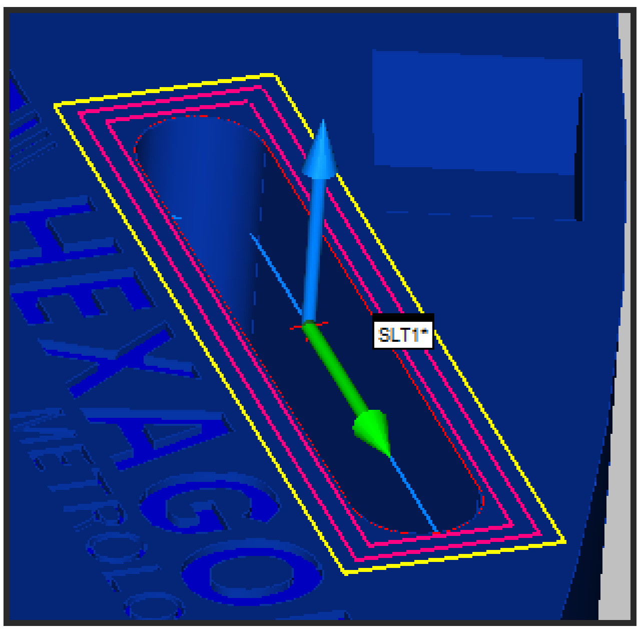
Some Sample Features with Overlays



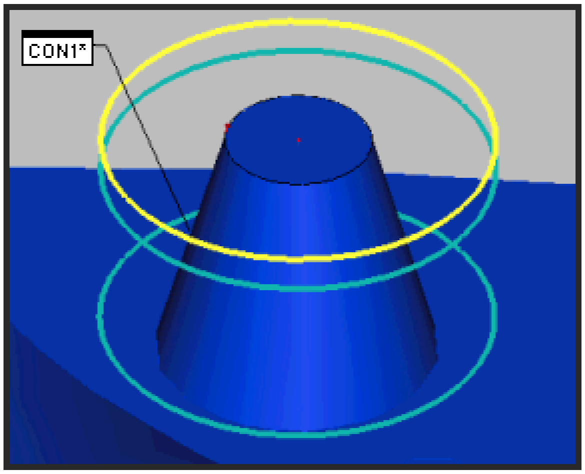
Sample Edge Point



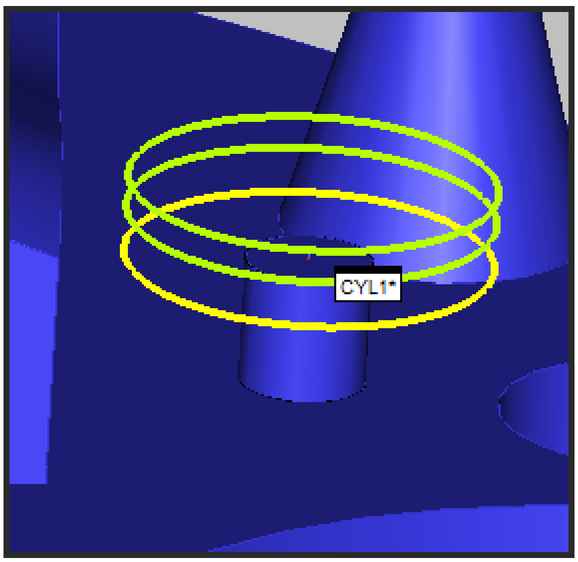
Sample Circle



Sample Slot



Sample DCC Cone



Sample Portable Cylinder

Pointcloud Scanning Colors

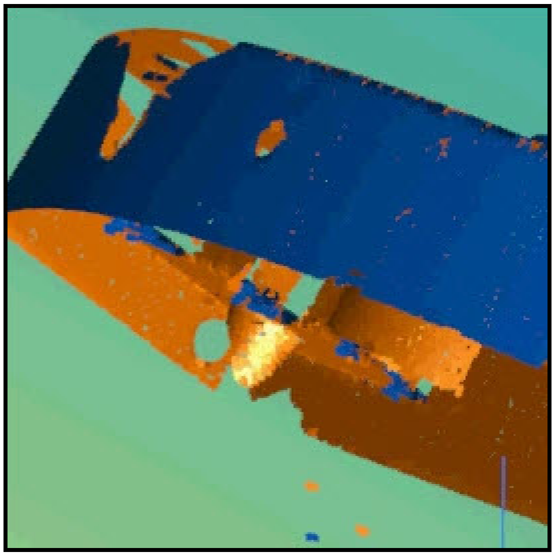
The following colors can help you interpret scanned pointclouds:

Blue - Existing scanned points of the outside of a part. Blue is the default outside color for a pointcloud. For information on how to change this color, see "Manipulating Pointclouds".

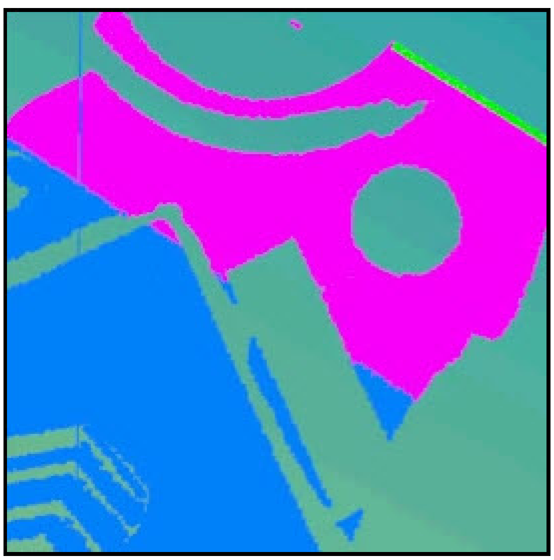
Orange - Existing scanned points of the inside of a part.

Magenta - Points currently being scanned.

Examples



Blue shows existing scanned points from the outside of a part. Orange shows the existing scanned points from the inside of a part.



Magenta shows the points currently being scanned.

Using the Laser Toolbars

In an effort to decrease the time it takes to program your part, PC-DMIS Laser offers you a variety of toolbars composed of frequently used commands. These toolbars can be accessed in two ways.

- Select the **View | Toolbars** submenu and select a toolbar from the menu provided.
- Right-click on PC-DMIS's **Toolbar** area and select a toolbar from the shortcut menu provided.

For a description of the standard PC-DMIS toolbars, please see the "Using Toolbars" topic in the PC-DMIS Core documentation.

The toolbars specific to Laser functionality are:

Pointcloud Toolbar

Pointcloud Toolbar

The **Pointcloud** toolbar provides all pointcloud operations, features and functions. It is accessible from the **View | Toolbars | Pointcloud** menu depending on your system's configuration.

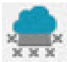
NOTE: All options may not be available. Some require specific licensing to enable them.

The following options are available from this toolbar:

Pointcloud button - Displays the **Pointcloud** dialog box used to create pointcloud features. For details on the dialog and creating pointcloud features, see the "Manipulating Pointclouds" topic in the "Using Pointclouds" PC-DMIS Laser chapter.

Pointcloud Operator button - Displays the **Pointcloud Operator** dialog box and is used to perform different operations on Cloud of Points (COP) commands and other Pointcloud operator commands. For details on the dialog and creating pointcloud operators, see the "Pointcloud Operators" topic in the PC-DMIS Laser documentation.

Pointcloud Mesh button - Displays the **Mesh Command** dialog box, used to define a mesh command for pointclouds. For details, see the "Creating a Mesh Feature" topic in the PC-DMIS Laser documentation. This option is only available if you have the Mesh and Big COP licenses.



Pointcloud Filtering Plane button - When clicked, the **Laser Data Collection Settings** dialog box is displayed. It is used to define data filtering and an exclusion plane for your pointcloud data. For details on the Pointcloud Filtering Plane, see "Laser Data Collection Settings" in PC-DMIS Laser documentation.

Pointcloud Boolean Operation button - Displays the **Pointcloud Operator** dialog box with the Boolean operator selected. For details on the dialog and creating a Boolean pointcloud operator, see the "BOOLEAN" topic in the "Pointcloud Operators" chapter of the PC-DMIS Laser documentation.

Cross Section Pointcloud button - Opens the **Pointcloud Operator** dialog box with the CROSS SECTION option selected in the Operator drop-down list.

Click the drop-down arrow to display the **Cross Section** toolbar:

For details on cross sections and using the **Cross Section** toolbar, see the "Cross Section" topic in the "Pointcloud Operators" chapter of the PC-DMIS Laser documentation.

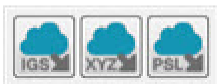
Clean Pointcloud button - When clicked, the CLEAN operation immediately eliminates outlier COP points based on the default MAX DISTANCE of the points to the CAD. If the distance of a point is greater than the value of MAX DISTANCE, the point is considered an outlier or not belonging to the part. To use this operation, you must have at least a rough alignment established (see "Creating a Pointcloud/CAD Alignment") and a CAD model. For details on the CLEAN pointcloud operator, see the "CLEAN" topic in the PC-DMIS Laser documentation.

Empty Pointcloud button - When clicked, PC-DMIS immediately removes all the data of the currently selected COP. Be aware that this change is permanent, so use with caution. For details on the EMPTY pointcloud operator, see the "EMPTY" topic in the PC-DMIS Laser documentation.

Filter Pointcloud button - Displays the **Pointcloud Operator** dialog box with the FILTER operation selected. The operation filters data to a smaller subset of points. For details on the FILTER pointcloud operator, see the "FILTER" topic in the PC-DMIS Laser documentation.

COP Export button - Displays the **Pointcloud Operator** dialog box for the currently selected export option.

Click the drop-down arrow to display the **COP Export** toolbar:



The options available are:

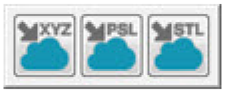
Export Pointcloud in IGES Format button - Displays the **Pointcloud Operator** dialog box with the EXPORT IGES operation selected. The Export IGES operation exports the data in a COP or operator command in IGES format to an IGES file. For details on exporting supported file types, see the "EXPORT" topic in PC-DMIS Laser documentation.

Export Pointcloud in XYZ Format button - Displays the **Pointcloud Operator** dialog box with the EXPORT XYZ operation selected. The Export XYZ operation exports the data in a COP or operator command in XYZ format to an XYZ file. For details on exporting supported file types, see the "EXPORT" topic in PC-DMIS Laser documentation.

Export Pointcloud in PSL Format button - Displays the **Pointcloud Operator** dialog box with the EXPORT PSL operation selected. The Export PSL operation exports the data in a COP or operator command in PSL format to a PSL file. For details on exporting supported file types, see the "EXPORT" topic in PC-DMIS Laser documentation.

COP Import button - Displays the **Pointcloud Operator** dialog box for the currently selected import option.

Click the drop-down arrow to display the **COP Import** toolbar:



The options available are:

Import Pointcloud in XYZ Format button - Displays the **Pointcloud Operator** dialog box with the IMPORT XYZ operation selected. The Import XYZ operation imports data from an external file into a COP command in the XYZ format. For details on importing supported file types, see the "IMPORT" topic in PC-DMIS Laser documentation.

Import Pointcloud in PSL Format button - Displays the **Pointcloud Operator** dialog box with the IMPORT PSL operation selected. The Import PSL operation imports data from an external file into a COP command in the PSL format. For details on importing supported file types, see the "IMPORT" topic in PC-DMIS Laser documentation.

Import Pointcloud in STL Format button - Displays the **Pointcloud Operator** dialog box with the IMPORT STL operation selected. The Import STL operation imports data from an external file into a COP command in the STL format. For details on importing supported file types, see the "IMPORT" topic in PC-DMIS Laser documentation.

Purge Pointcloud button - When clicked, PC-DMIS immediately removes all data points that do not belong to this operator. It is irreversible and affects all other operator commands that refer to same COP container so use with caution. For details on the Purge pointcloud operator command, see the "PURGE" topic in the PC-DMIS Laser documentation.

Reset Pointcloud button - When clicked, PC-DMIS immediately reverses the most recent Surface Colormap, Point Colormap, Select or Clean (unless Purge has been done) operations. For details on the Reset pointcloud operator command, see the "RESET" topic in the PC-DMIS Laser documentation.

Select Pointcloud button - Displays the **Pointcloud Operator** dialog box with the Select operator selected. This pointcloud operator provides by default the Polygon selection method. Select the vertices of the polygon then press the **End Key** to close it. For details on the Select pointcloud operator command, see the "SELECT" topic in the PC-DMIS Laser documentation.

NOTE: The **Select Pointcloud** option differs from the use of the pointcloud operator as it only applies the function and is not added as a command. To create the command, open the pointcloud operator and choose the **Select** method.

TCP/IP button - Performs the currently selected operation described below.

Click the drop-down arrow to display the **TCP/IP** toolbar:

The options available are:

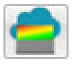
TCP/IP Pointcloud Server Connection with Local Copy button - This establishes the connection with the client, sends the point cloud data directly to the client, and when the scan finishes, the point cloud data remains inside the measurement routine. For details on TCP/IP Pointcloud server connection, see the "TCP/IP Pointcloud Server" topic in PC-DMIS Laser documentation.

TCP/IP Pointcloud Server Connection without Local Copy button - This establishes the connection with the client, sends the point cloud data directly to the client, and when the scan finishes, the point cloud data is deleted from the measurement routine. For details on TCP/IP Pointcloud server connection, see the "TCP/IP Pointcloud Server" topic in PC-DMIS Laser documentation.

Pointcloud Alignment button - When clicked, the **Pointcloud/CAD Alignment** dialog box appears. It is used to create Pointcloud to CAD and COP to COP alignments. See the "Alignment Dialog Box Description" topic in the "Pointcloud Alignments" chapter of the PC-DMIS Laser documentation.

Pointcloud Point Colormap button - Displays the **Pointcloud Operator** dialog box with the Point Colormap operator selected. The Point Colormap operation evaluates the deviations of the data points contained in a COP command compared to a CAD object.

For details on the Pointcloud Point Colormap operator, see the "POINT COLORMAP" topic in PC-DMIS Laser documentation.

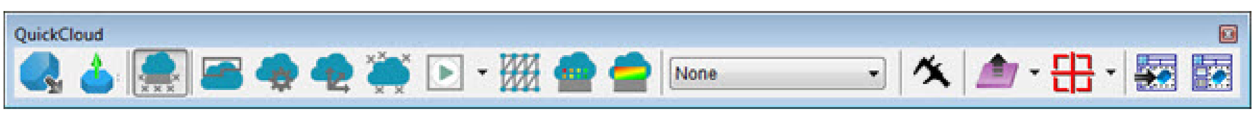


Pointcloud Surface Colormap button - Displays the **Pointcloud Operator** dialog box with the Surface Colormap operator selected. The SURFACE COLORMAP operation applies a colored shading to the CAD model. The model is shaded according to the deviations of the cloud of points compared to CAD using the colors defined in the **Edit Dimension Color** dialog box and the tolerance limits specified in the **Upper tolerance** and **Lower tolerance** boxes For details on the Pointcloud Surface Colormap operator, see the "SURFACE COLORMAP" topic in PC-DMIS Laser documentation.

You can create multiple surface colormaps in a PC-DMIS measurement routine. Only one is active at a time however. The last surface colormap that was applied and created, or the last one executed, is always the currently active colormap. You can also select which colormap is the active one using the Surface Colormap list box. When a new colormap is activated, its associated scale with tolerance values and any annotations are displayed in the Graphic Display window.

To do this, click the **SurfaceColorMaps** list box and select the colormap from the list of defined Surface Colormap operators:

QuickCloud Toolbar



QuickCloud toolbar

The **QuickCloud** toolbar is only available when PC-DMIS is licensed and is configured as a Portable device. It provides the buttons to complete all the steps from beginning to end for working with COP.

For detailed information on this toolbar, see the "QuickCloud Toolbar" topic in the "PC-DMIS Portable" documentation.

NOTE: For details on all Pointcloud toolbar functions, see the "Pointcloud Toolbar" topic in PC-DMIS Laser documentation.

Using Pointclouds

The Cloud of Points command (COP) allows you to store XYZ coordinate data that can come directly from a laser sensor through one or more referring scan commands. You can also input data directly into a COP from other PC-DMIS features or external data files.

You can add Pointclouds to your measurement routine in these ways:

- Select the **File | Import | Pointcloud** sub menu and select a data file to import (*XYZ*, PSL, or *STL*).

  STL: The STL file type is the same file type that is covered in the "Importing an STL File" topic of the Core PC-DMIS documentation, except that instead of importing the file as a CAD model, it imports the file as a pointcloud.

  XYZ: The XYZ file type is the same file type that is covered in the "Importing an XYZ File as CAD Data" topic of the Core PC-DMIS documentation, except that instead of importing the file as a CAD model, it imports the file as a pointcloud.

- Select the **Insert | Pointcloud | Feature** menu item to open the **Pointcloud** dialog box.
- Manually type the COP command into the Edit Window. Press **F9** on the COP command in the Edit Window to open the **Pointcloud** dialog box. For information on the COP command mode text, see "COP Command Mode Text".
- From the **Pointcloud** toolbar, click the **Pointcloud** button  to open the **Pointcloud** dialog box.

For information on how to manipulate Pointclouds from the **Pointcloud** dialog box, see the "Manipulating Pointclouds" topic.

PC-DMIS uses additional, laser sensor-related commands and tools that support Pointcloud functionality. They are:

- Pointcloud Operators
- Pointcloud Alignments
- Pointcloud Point Information
- Laser Data Collection Settings

IMPORTANT: Your license or port lock must contain a license with either the **Small COP (COP)** or **Big COP** options to use COP capability.

About the Small COP (COP) and Big COP Laser Options

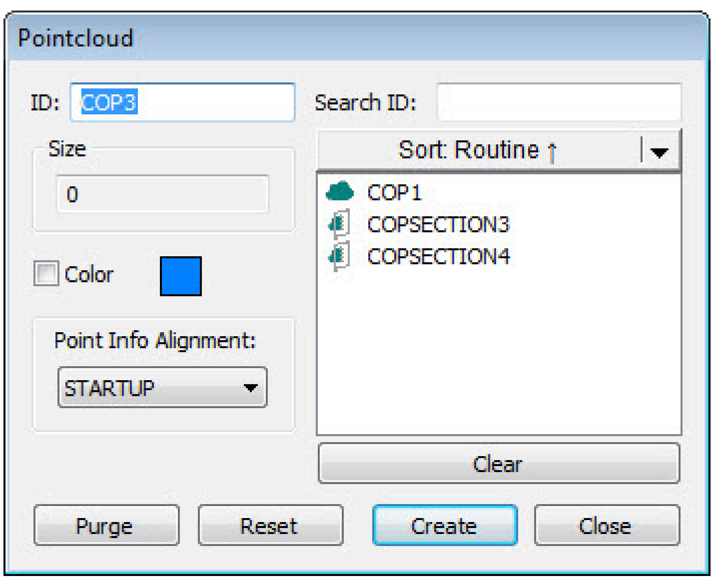
The PC-DMIS CAD++ license includes the **Small COP (COP)** option. It provides limited pointcloud functionality.

The PC-DMIS Laser option (but not including Vision probes), includes the **Big COP** option. This option provides full pointcloud functionality. You can purchase it separately for other configurations.

The following describes the differences in functionality between the **Small COP (COP)** and **Big COP** licensing options:

- If **Small COP (COP)** is enabled and **Big COP** is disabled, PC-DMIS limits the pointcloud size to 500,000 points. The pointcloud automatically re-sizes to stays within the limit.
- Pointcloud alignment is enabled only when **Big COP** is enabled.
- Meshing is enabled only if both **Big COP** and **Mesh** are enabled
- If **Small COP (COP)** and **Big COP** options are disabled, pointcloud functionality is disabled

Manipulating Pointclouds



Pointcloud dialog box

NOTE: The **Pointcloud** dialog box only has an effect if the COP command contains data.

The **Pointcloud** dialog box is accessed by clicking the **Pointcloud** button from the Pointcloud toolbar or clicking **Insert | Pointcloud | Feature** from the menu.

The dialog box contains the following elements:

ID - Contains a unique identity of the pointcloud command being edited.

Search ID - If there's a long list of operators defined, you can search using the **Search ID** field to locate specific operators in the list. Begin entering the operator's ID into the field and the list automatically filters based on the entry.

Size - Total number of points in the pointcloud.

Color - Sets the color for the scanned points in the pointcloud on the outside of a part. To change the pointcloud color, select the **Color** check box and then click on the **Color** box to select the needed color from **Color** dialog box. For additional information on pointcloud colors see, "Pointcloud Scanning Colors".

Commands List - This area contains the list of features or scans that send data to the COP command in the dialog box. A **Sort** functionality is available to organize the list by **ID**, **Type**, **Routine** or **Time**. Select the option from the drop-down list then click the **Sort** button.

Point Info - With the **Pointcloud** dialog box open, clicking on a pointcloud point in the graphics window opens the **Pointcloud Point Information** dialog box containing information about the point with respect the alignment. This box contains the point's numerical ID, its coordinates, and estimated normal of the point. Corresponding CAD points are also displayed with CAD coordinates and CAD normal. Finally, the deviation between the Point and CAD is shown with the scale for the deviation arrow specified in the dialog. Point selection has no associated operator command. With the **Pointcloud Point Information** dialog open and clicking on the **Create Point** button, two scenarios are possible:

- If there is a CAD model in the measurement routine and the pointcloud is aligned, a **Laser Surface Point** is created, inserted and resolved at the selected position.
- Otherwise, a **Constructed Offset** point is created and inserted in the measurement routine.

Purge / Reset - The **Reset** button restores all the data stored in a COP command. The **Purge** button permanently deletes all the data in a cloud of points that is not currently shown, selected, or filtered. This causes the cloud of points to only keep the visible data.

See "Pointcloud Point Information" for information on viewing Pointcloud point deviation information.

COP Command Mode Text

The COP command inside the Edit window's Command Mode looks like this:

```
COP1 =COP/DATA,SIZE=0
    REF,,
```

The COP command must precede any scan referring to it in the measurement routine.

EXAMPLE: For example, `REF,SCN2` shown below points to the `SCN2` scan and uses its data:

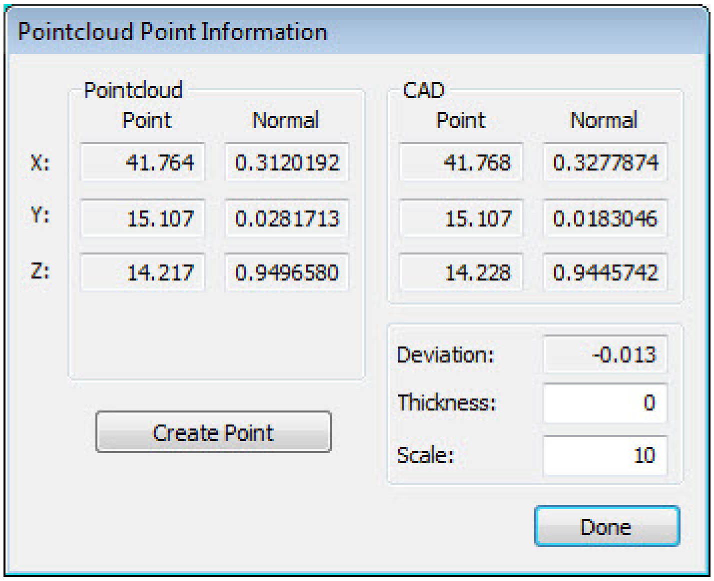
```
COP2 =COP/DATA,SIZE=0

REF,SCN2,,
```

NOTE: You can have more than one scan refer to a COP command.

Be aware that if you cut a COP command and paste it again, the resultant command gets pasted without the data points. If you need to move your COP command to a different location in the Edit window, you need to re-create the COP command at the desired location and delete the earlier one.

Pointcloud Point Information

With the **Pointcloud** dialog box, you can view point-specific information. To do this, on the cloud of points (COP) in the Graphic Display window, click on a point. This opens the **Pointcloud Point Information** dialog box.



Pointcloud Point Information dialog box

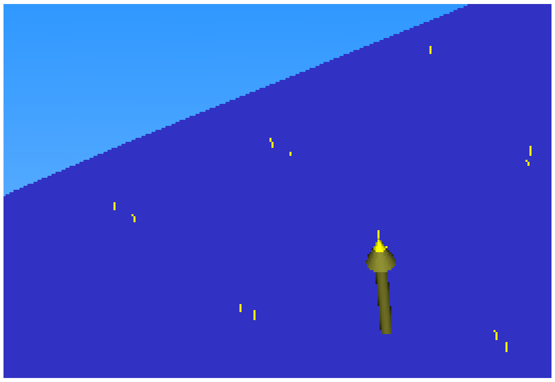
From this dialog box you can view the **XYZ** and the **Normal** point vector values for the Pointcloud point, as well as the **ID** for the selected point. It also shows the corresponding CAD's **XYZ** and **Normal** vector values.

Deviation - Displays the distance from the Pointcloud point to the corresponding CAD point.

Thickness - The software adds this value to the deviation from the CAD value that it computes when you click on a Pointcloud point. This value is useful, for example, if you have a CAD surface model and you want to add a material thickness.

Scale - This value determines the scale that the deviation arrow uses in the Graphic Display window. For example, A scale of 10 would display an arrow with a length that is ten times the length of the deviation.

The deviation arrow appears when you select a point from the Graphic Display window. The arrow indicates the direction of the point deviation from the CAD.



Point Deviation Arrow

Create Point button - This creates a constructed offset point for the selected point. The software names the constructed offset point with the following convention and then adds the point to the measurement routine: **<pointcloud name>_P<point ID>** (for example, COP1_P185048).

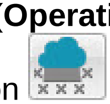
> **HINT:** If you use a laser sensor when you click **Create Point**, the software creates a laser surface point instead of a constructed offset point.

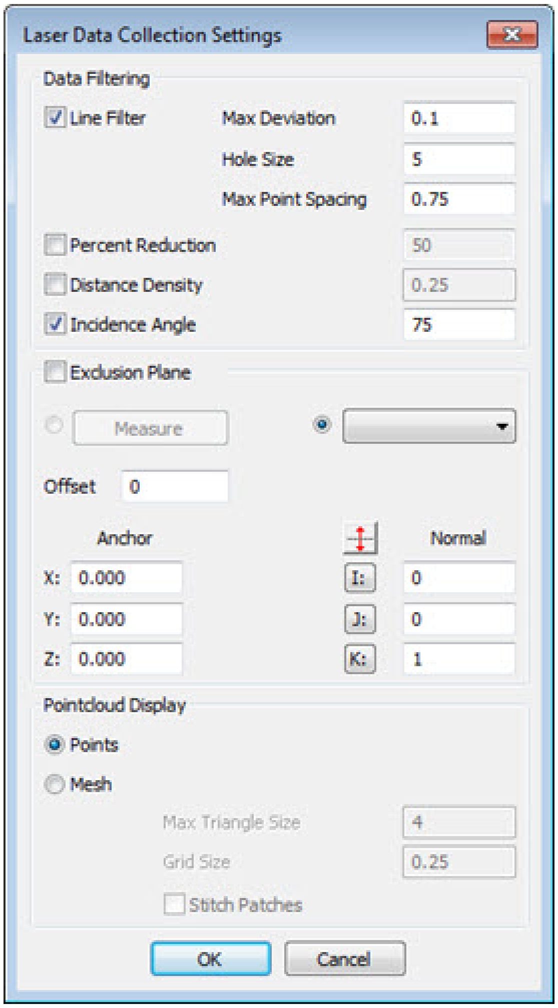
Constructed Point from Pointcloud

Using Point Data for Auto Features

With the **Auto Feature** dialog box open, you can click on desired points from the pointcloud to provide input data for a given auto feature. See "Auto Feature Extraction" for more information.

Laser Data Collection Settings

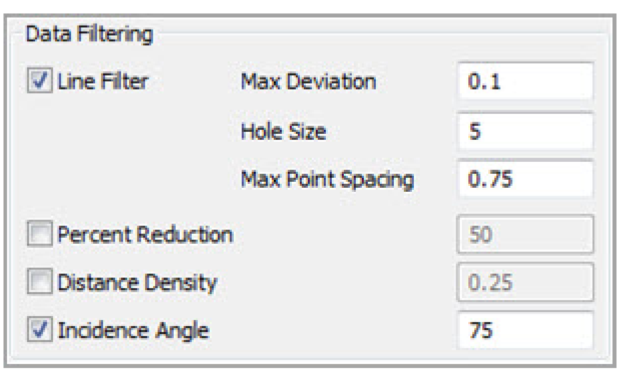
Access the **Laser Data Collection Settings** dialog box (**Operation | Pointcloud | Data Collection** or click the **Pointcloud Filtering Plane** button  on the Pointcloud or QuickCloud toolbar).



Laser Data Collection Settings dialog box

The **Laser Data Collection Settings** dialog box allows you to define the types of data filtering, the exclusion plane and the pointcloud display for laser scanned data.

Data Filtering Section



Data filtering allows real-time filtering of the data. It removes the data as you scan.

The **Data Filtering** section provides the following options:

Line Filter: A real-time filter for individual lines, which provides smoothing and point reduction of incoming data from the laser sensor.

Mark the **Line Filter** check box to enable these options:

Max Deviation: As each incoming scan line is evaluated, points may be moved or smoothed in relation to their neighboring points. This setting defines the maximum allowed value that a point may be moved or smoothed.

Hole Size: When the software evaluates a scan line, and it detects a hole or gap of the specified size (or larger), the filter treats the scan segments as separate lines. In most cases this can be set to the size of the smallest hole on the physical part.

Max Point Spacing: When analyzing the incoming scan data and reducing the number of points, this setting defines the maximum distance between two consecutive points. If the scan surface is curved, the resulting point spacing is typically smaller than the **Max Point Spacing** value.

When this parameter is set to zero, no point reduction is done. Typically this value should be set to less than 1/3 of the Hole Size.

The **Max Point Spacing** setting determines the resolution of the scanned points. For most parts, the default values listed below can be used. To obtain higher resolution when scanning parts with small details, a smaller **Max Point Spacing** may be used. Using a smaller **Max Point Spacing** results in fewer points being filtered out and increases the total COP size.

	Max Point Spacing
Large Details	1 mm / 0.03937 inch
Default	**0.75 mm / 0.02953 inch**
Small Details	0.5 mm / 0.01968 inch
Fine Details	0.25 mm / 0.00984 inch

Percent Reduction: Removes a percentage of the pointcloud data collected.

1. Select the **Percent Reduction** option and in the box to its right, type a percentage value between 0-100. The value is the percent of the collected

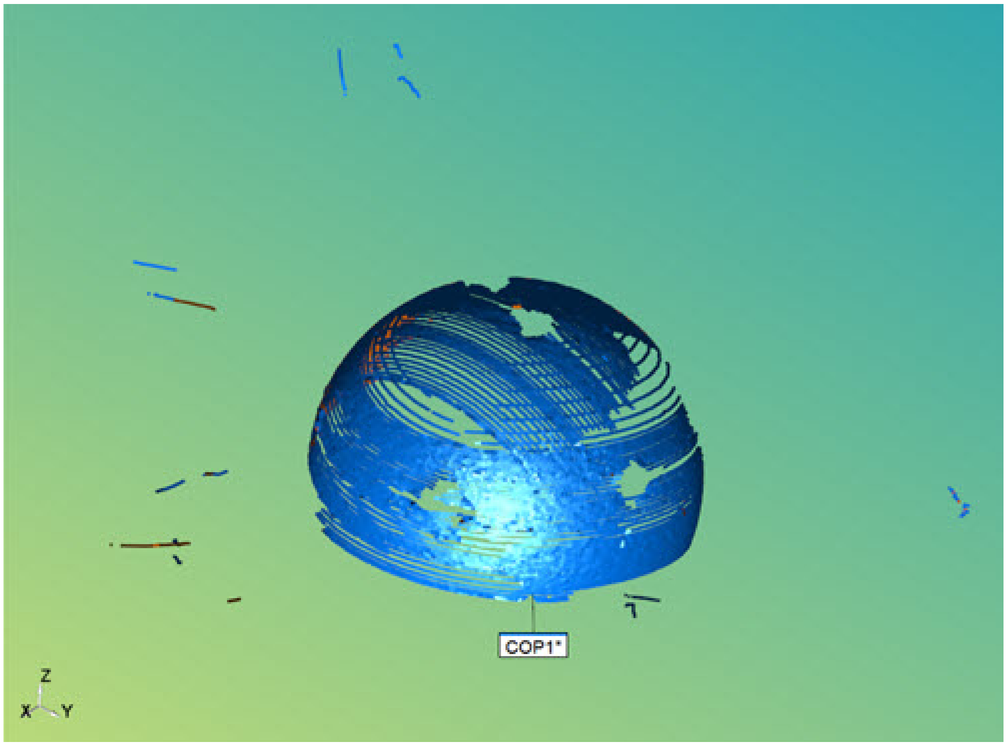
pointcloud data you want the software to filter out. If you enter zero, no filtering takes place.

2. Click **OK** to apply this to your measurement routine.

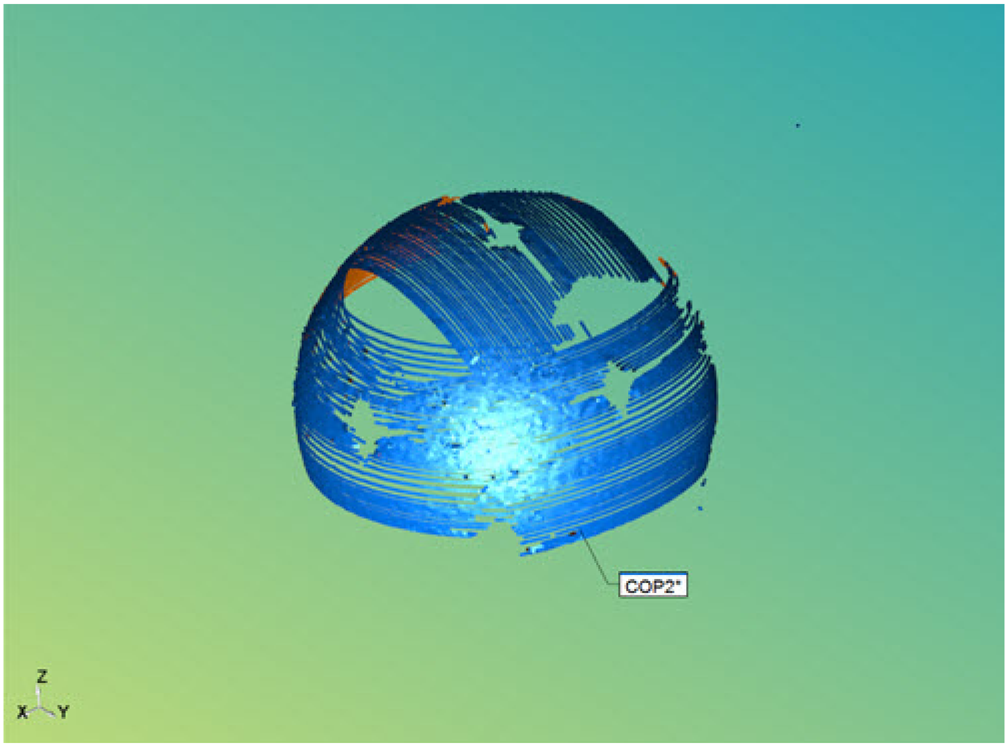
Distance Density: Provides a filtering based on the point distance value. If the distance between a point and its neighboring points is less than this value, the software discards the point. This option becomes available if you select the **Points** option in the **Pointcloud Display** section of the dialog box.

1. Select the **Distance Density** option and in the box to its right, type a distance value in the measurement routine units. Values greater than or equal to zero are valid. 1 mm is the default value - if your measurement routine uses inches, the software converts 1 mm to inches.
2. Click **OK** to apply the filtering.

Incidence Angle: Filters out all scanned points that have an incident angle greater than the entered value. The **Incidence Angle** check box is marked by default with a default value of 75. The angle is calculated between the estimated surface normal and the scan direction of the laser sensor. The smaller the value, the more points are filtered out.



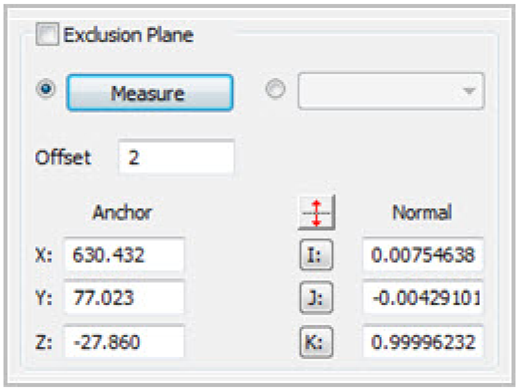
Shiny sphere with no incidence angle applied



Shiny sphere with incidence angle on at default value 75

The **Incidence Angle** filter can be applied real-time, while scanning. During scanning, the angle of the scan line relative to the measured surface is determined; any points outside the specified angle are automatically removed and discarded.

Exclusion Plane Section



You can use exclusion planes to remove all points within the defined area of the plane. To enable this feature, mark the **Exclusion Plane** check box.

When the **Exclusion Plane** check box is marked, the software activates the defined exclusion plane. If the icon on the toolbar is in the pushed state, filtering is enabled.

Once activated, the software uses the exclusion plane the next time the measurement routine is executed.

HINT: You can tell when the Exclusion Plane is active in your measurement routine by how the **Pointcloud Filtering Plane** button appears on the QuickCloud or Pointcloud toolbars. If the button appears to be pressed in, the exclusion plane is active, otherwise it is not active.

There are three ways to define the exclusion plane:

- **Measure**

  Use a contact probe or laser sensor to measure the exclusion plane.

  Click the **Measure** button, then take three hits with a contact probe to measure the exclusion plane. With a laser sensor, scan the area of the plane. If an alignment already exists, the plane is automatically defined in that alignment. If not, the plane is defined using the machine coordinates. If that changes, you need to redefine the plane.

- **Entering the XYZ and IJK values**
- You can also define the exclusion plane by its normal vector and an anchor point. The exclusion plane is independent of the data filtering.

  To define an exclusion plane:

  1. Edit the XYZ anchor positions if necessary.
  2. Click the **I**, **J** or **K** normal button that your plane is relative to and edit the value if necessary. To automatically change the direction of the normal value, you can click the **Reverse Direction** button.
  3. If you are in online mode, you can click the **Measure** button to measure your defined exclusion plane.
  4. Click **OK** to save your settings.

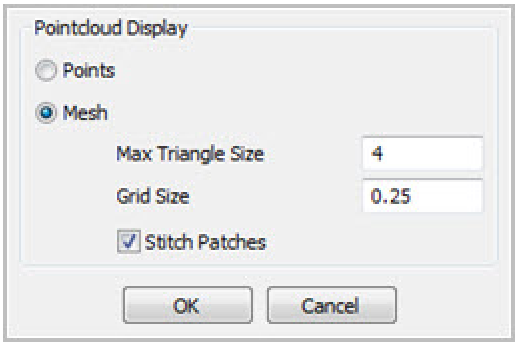
- **Select an Existing Plane**

  Select an existing plane (a plane which already exists in the measurement routine) from the **Exclusion Plane Feature** list. The Anchor and Normal (vector) fields update accordingly.

  By selecting an existing plane, when the measurement routine is re-executed and the plane is re-measured, this becomes the new exclusion plane used for the COP. This is useful for portable devices, if the device is moved or if the part is moved to a different surface.

Offset - This offsets the plane in the defined Normal direction by the value entered (in measurement routine units).

Pointcloud Display Section



The **Pointcloud Display** section allows you to display the pointcloud as points or as a mesh when performing laser scans. It facilitates the identification of areas not covered with data.

Points - This option displays the pointcloud as a set of points. The **Distance Density** filter in the **Data Filtering** section of the dialog box is enabled when this option is selected. It is used to define valid point distances for the points used to create the pointcloud.

Mesh - This option causes laser data to appear as a mesh during scanning. The software displays the current scan pass as a pointcloud and previous passes as a mesh. This option is only available for Portable systems.

HINT: The Mesh display is relative to the orientation of the laser sensor. While scanning, if the laser sensor orientation changes more than 25 degrees in a single scan pass, the software meshes the collected data, and automatically creates a new scan.

The **Max Triangle Size** and **Grid Size** values define the displayed mesh. After you scan, the software displays the data as a mesh until you close and reopen the measurement routine. The data then appears as a pointcloud. The mesh display functionality requires the Mesh license.

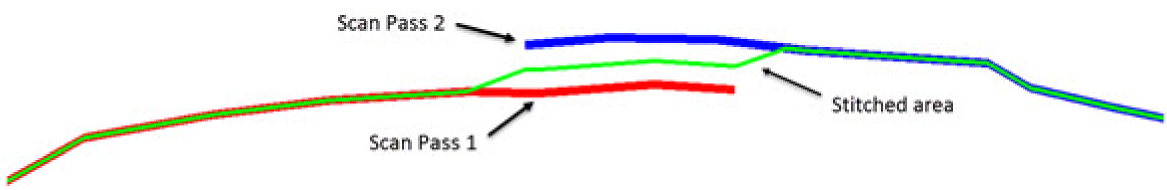
- If scanning speed is slow, and more than one point is in a grid square, PC-DMIS keeps the best point.
- If scanning speed is fast, it is possible to have a grid square without any data, which may cause gaps in the displayed mesh.

Max Triangle Size - This value determines the largest possible triangle in the mesh display. If the distance between any two points is greater than this value, the software does not create any triangles. If there are hole features on your part, you typically need to set this value to be slightly smaller than the smallest hole. This prevents the mesh from filling the hole.

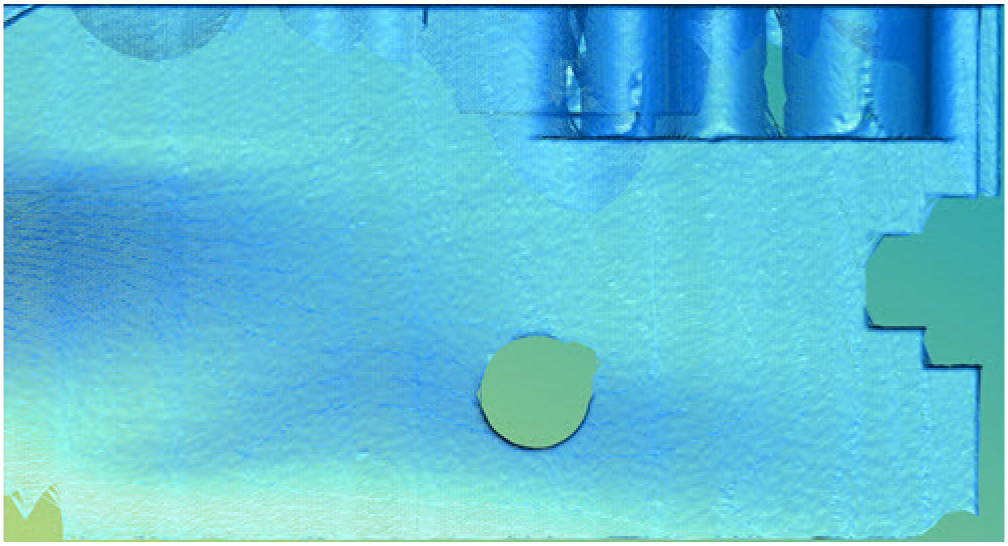
The default value for the **Max Triangle Size** is 5mm. The software converts this to inches if your measurement routine is using that unit. Valid range values depends on the size of the part.

Grid Size - This value defines the size of the triangles used to create the mesh. This value also affects the mesh's resolution and how refined it appears. The smaller the value, the more time it takes to generate the mesh but the higher the resolution of the resulting mesh. Be aware that this value is critical; it can affect the speed of data collection if it is set too small.

Stitch Patches check box - When scanning as **Mesh** display and the **Stitch Patches** check box is marked, multiple scan passes are blended and the overlapping data is removed.

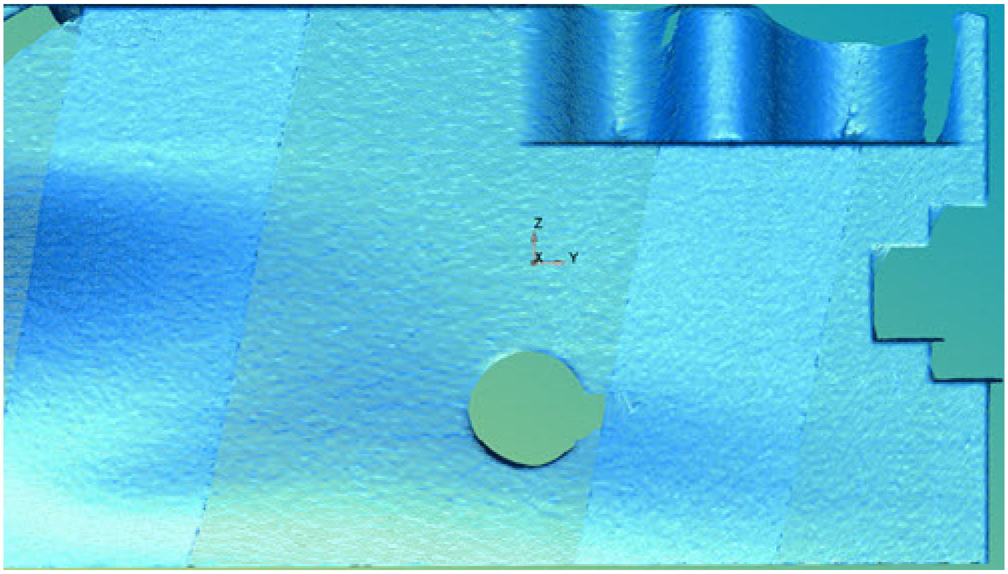


The overlapping scan passes must be within a distance lower than the point density in order to be stitched.



Example of Stitch Patches turned ON when scanning as Mesh display

When scanning as **Mesh** display and the **Stitch Patches** check box is NOT marked, multiple scan passes are overlayed on top of each other.



Example of Stitch Patches turned OFF when scanning as Mesh display

To use this feature:

1. From the **Pointcloud Display** section of the dialog box, click **Mesh**.
2. In the **Grid Size** box, type the value to define the mesh triangle size. A recommended starting value is 0.25mm (at 1/64 inches). A smaller grid size provides a smaller (higher quality) resolution when creating the mesh.
3. If the distance between any two points is greater than the **Max Triangle Size** value, the software does not create any triangles. If there are hole features on your part, you typically need to set this value to be slightly smaller than the smallest hole. This prevents the mesh from filling the hole.
4. Click **OK** to finish.

Using the Simulate Pointcloud Function

The **Simulate Pointcloud** function allows you to create and view the pointcloud from the **Scan** dialog box (linear, freeform, and so on) when the CMM is in offline mode. This way you can see if the simulated pointcloud is acceptable and make changes if needed for an individual scan. PC-DMIS holds the simulated points in a COP.

Follow the "Getting Started" chapter to define the active sensor tip and scan speed. If you like, you can pre-define the laser width and density of the scan from the **Measure Laser Probe** dialog box when you define the sensor. For details, see the "Measure Laser Probe Options" topic.

Define the scan path properties from any **Scan** dialog box (linear, freeform, and so on). You can also define the laser width and density settings from the same dialog box. For details, see the "Scan Zoom States (for CMS Sensors)" topic.

Click the **Simulate** button to display the pointcloud in the Graphic Display window.

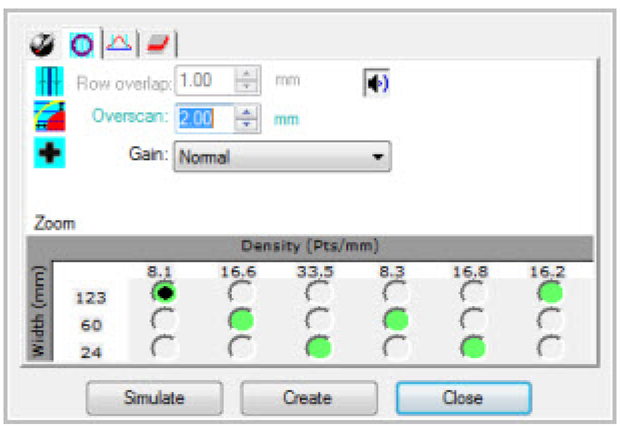
After you create the scans, you can execute the entire offline measurement routine and display all scans in different probe orientations. This allows you to check if you can extract scanned Auto Features (for example) based on the scan settings.

Warning: If the CMM is online and the **Simulate** button is pressed on the **Laser Scan** dialog box (Freeform, Linear Open, etc.), the software immediately drives the machine and scans online. To prevent injury, make sure you are clear of the machine prior to pressing the button.

Example Using the Simulate Pointcloud Function

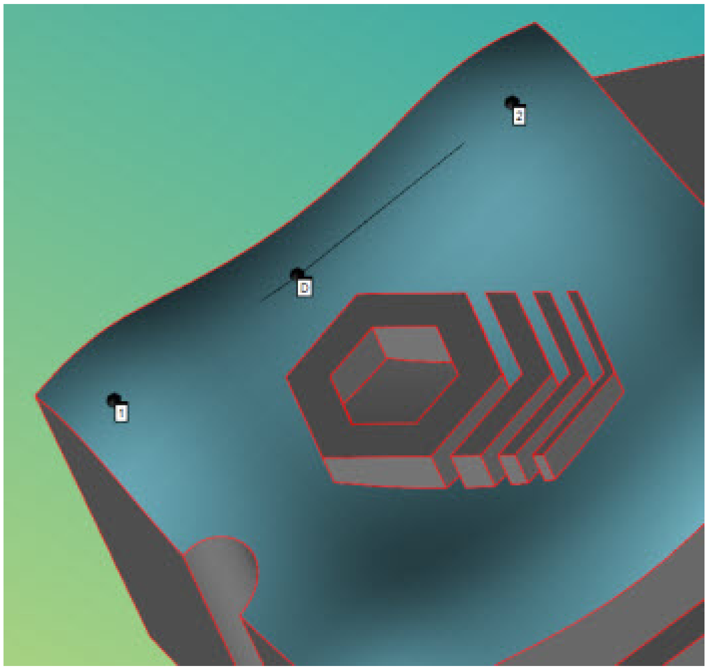
For example, to use the Simulate pointcloud function on a linear open scan:

1. Create a COP (**Insert | Pointcloud | Feature**). For details on pointcloud features and creating a COP, see the "Using Pointclouds" chapter.
2. Set the scan speed. For details, see the "Getting Started" topic.
3. Open the **Linear Open Scan** dialog box (**Insert | Scan | Linear Open**).
4. In the **Scan Properties** section, set the **Increment** value as necessary.
5. At the bottom of the dialog box, click the **Laser Scan Properties** tab; and then set the **Overscan** value, select the appropriate **Gain** option, and the stripe width/scan density as needed.



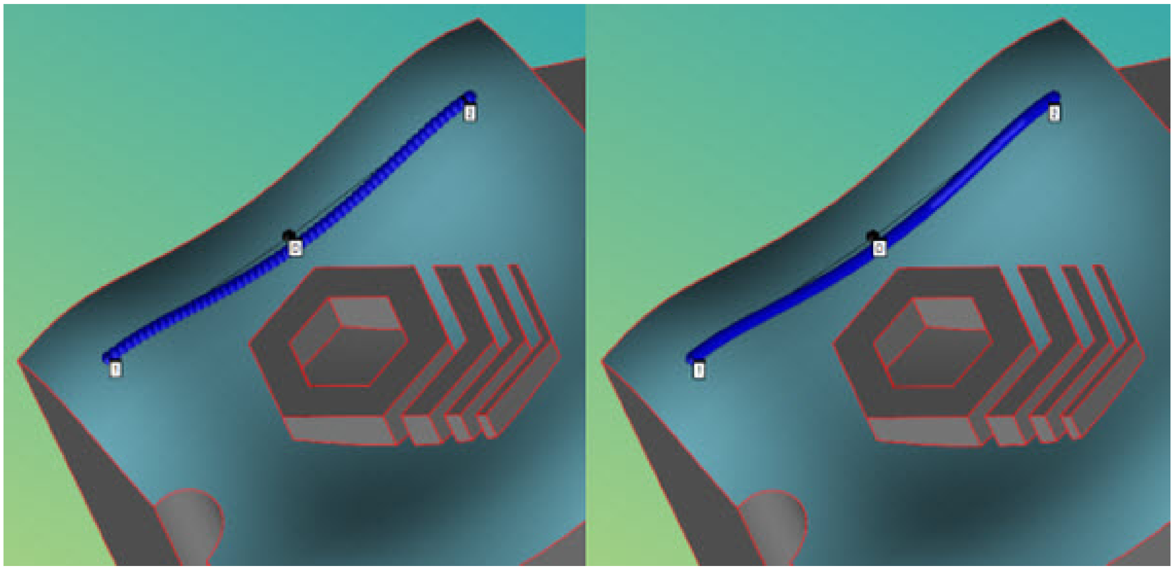
Laser Scan Properties tab

6. In the Graphic Display window, click the three points on the CAD model to define the boundary points and vectors as usual.



Example showing the three points to setup the scan

7. From the **Boundary Points and Vectors** section, click the **Generate** button. For an example, see the result on the left image below.
8. From the **Theoretical Scan Points** section, click the **Spline Points** button. For an example, see the result on the right image below.



Example showing a Linear Open scan generated (left) and splined (right)

9. Click the **Simulate** button to show the simulated pointcloud based on the current probe orientation (active tip) and laser scan settings.

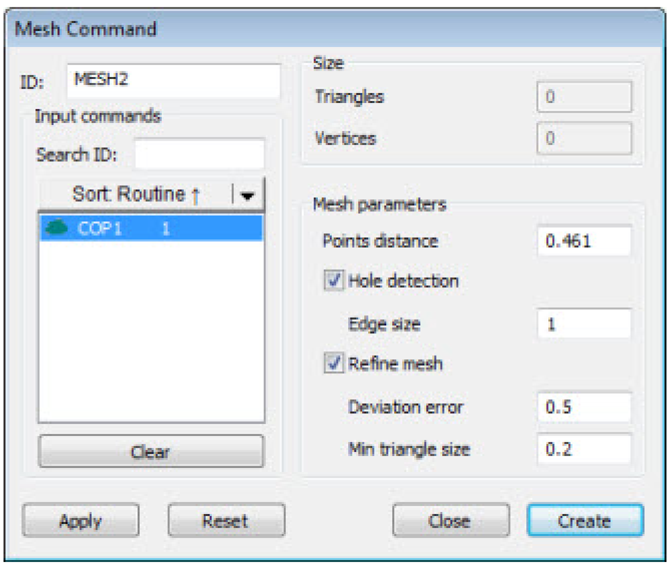
If needed, you can make changes to the scan and simulate to check the results.

10. When everything looks correct, click the **Create** button to implement the scan in your measurement routine.

Creating a Mesh Feature

The Mesh command is used to create a mesh from any number of pointclouds. You need to have one or more COPs already created in order to perform the Mesh operation. For details on creating pointclouds, see the "Using Pointclouds" chapter.

Select **Insert | Pointcloud | Mesh** from the main menu to display the **Mesh Command** dialog box. You can also access the dialog box by clicking the **Pointcloud Mesh** button from the Pointcloud or QuickCloud toolbars.



Mesh Command dialog box

The **Size** section details the number of triangle and vertices defined in your Mesh feature.

To create a Mesh feature:

1. Select the features and pointclouds to be meshed together from the list.
2. Update the options in the **Mesh Parameters** section as needed:

- **Points Distance** - The minimum distance between neighboring points used to create the vertices of each triangle in the mesh.
- **Hole Detection** check box - When marked, PC-DMIS determines when to exclude points based on the **Edge Size** value.

  - **Edge Size** - The value entered is used to determine when two points of the pointcloud are going to be included in the Mesh being created. If the distance is greater than the **Edge Size** value, it's considered a hole and the point is excluded. A value of -1 defines an edge size without limit.

- **Refine Mesh** check box - When marked, the following parameters are used to refine the mesh being created:

  - **Deviation Error** - The value entered determines how far points can deviate from the mesh construction and still be included in the mesh.
  - **Min Triangle Size** - The value entered determine the minimum size a triangle can be based on the points being evaluated.

3. Click **Apply** to apply any changes made in the **Mesh Command** dialog box. Click **Create** to generate the new Mesh command.

Click the **Reset** button to remove the created mesh from the Edit and Graphic Display windows.

Click the **Close** button to close the mesh dialog box and cancel the mesh operation if the **Create** button wasn't clicked.

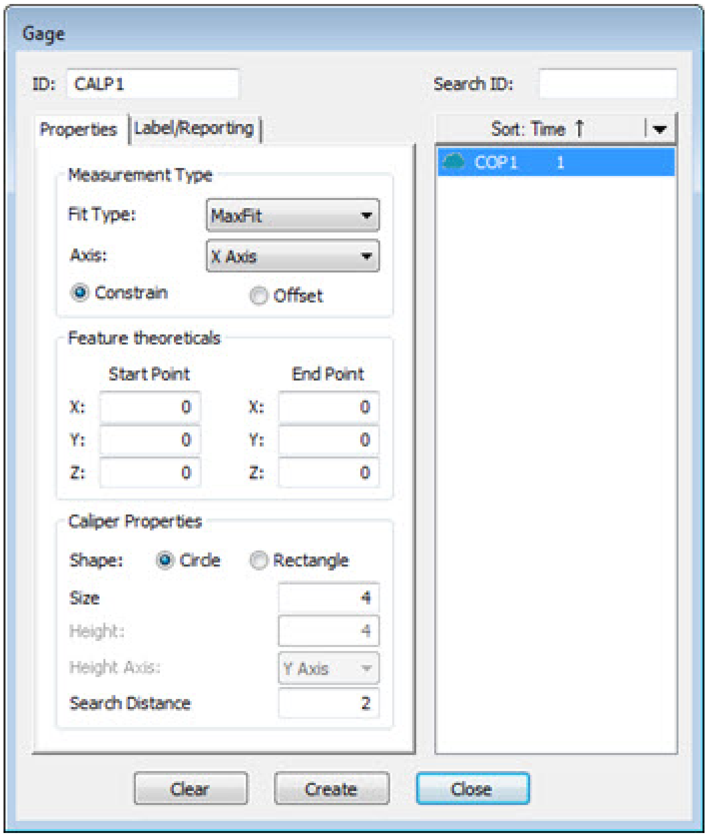
Caliper Overview

IMPORTANT: This option is only available if your PC-DMIS license includes the Small COP or BigCOP option.

The **Caliper** is a quick check tool which works similar to a physical caliper. It provides a local two point size check on the Pointcloud (COP), Mesh or COPOPER (such as the COPSELECT, COPCLEAN, or COPFILTER) object. The Caliper shows the measured length along the selected axis or direction.

Select the **Caliper** option from the **Insert | Gage** menu.

HINT: You can also click the **Caliper** button  from the **QuickCloud** or **QuickMeasure** toolbar to open the **Gage** dialog box.

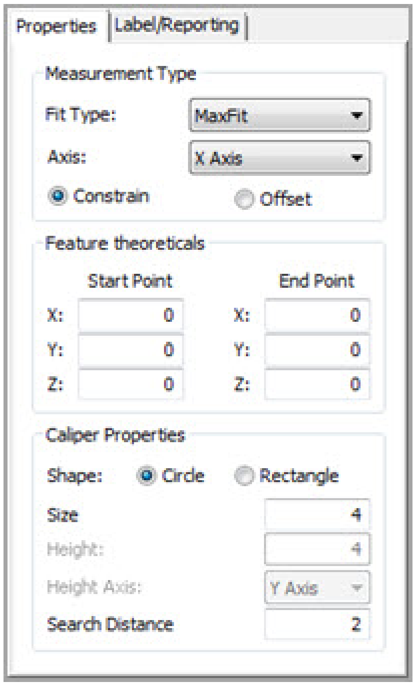


Gage dialog box

A **Caliper** has two tips, which are used to measure the distance between two opposing sides. The caliper tip has a user-defined size. Click in the Graphic Display window to select the start and end points. Using the data within the tip size, the caliper endpoints stop at the high points on the selected data (or optionally on the calculated best fit points). The software performs a search distance along the caliper axis to determine the relevant points.

The **Gage** dialog box has these tabs:

Gage dialog box - Properties tab



Gage dialog box - Properties tab

The **Properties** tab of the **Gage** dialog box has these sections:

Measurement Type

Fit Type: Click the drop-down arrow to display these options:

MaxFit: This is the default setting. Using the tip size and search distance, the caliper endpoints stop at the high points on the selected surfaces. A search distance along the caliper axis is used to determine the relevant points.

BestFit: A best fit least square fit is applied to all data points that fall within the caliper tip size and search distance. The resulting best fit points are used to determine the caliper length. This alternative method may be used if the scan data contains "noise", but can result in the caliper shown inside the pointcloud or mesh.

Axis: The Caliper can be constructed along the X, Y, or Z axis. Select **Parallel** to construct it Normal to the first surface picked. Select **None** to apply no constraint (3D distance between two points).

Constrain: Select this option and the two endpoints are exactly opposite one another, along the selected axis.

Offset: Select this option and the two endpoints can be offset from each other in position, but the measured length remains along the selected axis.

Feature theoreticals

Start Point: This is the XYZ coordinate location of where the Caliper begins.

End Point: This is the XYZ coordinate location of where the Caliper stops.

Caliper Properties

Shape: Select the appropriate tip shape, **Circle** (default) or **Rectangle**. If you select **Rectangle**, the **Height** and **Height Axis** options are enabled.

> **NOTE:** The **Rectangle** option is only enabled when you select the **X Axis**, **Y Axis**, or **Z Axis** option from the **Measurement Type** section. If you select **Parallel** or **None**, the **Rectangle** option is disabled.

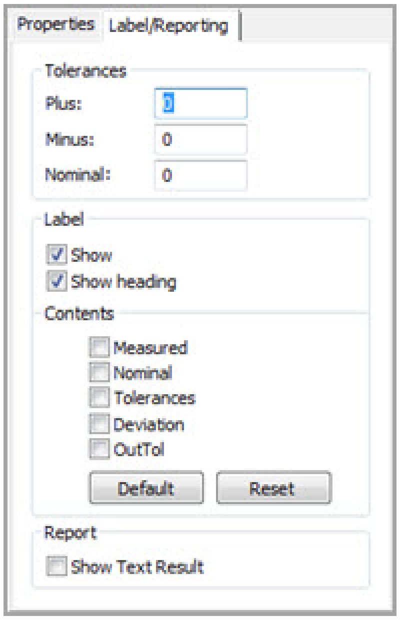
Size / Width: Enter the **Size** for **Circle** tip type (or **Width** if you select the **Rectangle** tip type) of the tip used. This value defines the caliper tip size (or rectangle tip width). When the distance is then computed, the tip stops at the high point the way a caliper would.

Height: This value defines the rectangular probe height. The height size runs along the selected axis.

Height Axis: Select the option from the list to set the axis used to control the rotation of the rectangle.

Search Distance: This value defines the length, from nominal, on either side of the picked point. The search distance along with the caliper tip shape creates a cylindrical zone. All data within this zone is evaluated to determine the caliper high point.

Gage dialog box - Label/Reporting tab



Gage dialog box - Label/Reporting tab

The **Label/Reporting** tab of the **Gage** dialog box has these sections:

Tolerances section

NOTE: The default Caliper tolerances are defined by the Dimension Color scale. For details, see "Editing Dimension Colors" in the Core help documentation.

The Tolerances section allows you to type plus and minus tolerances for the Caliper length.

To enter the plus, minus and nominal tolerances:

1. Type in the plus tolerance in the **Plus** box
2. Type the minus tolerance value in the **Minus** box

If a CAD model is used, the nominal (theoretical) caliper length is determined from the CAD. If no CAD model is used, the nominal value is updated with the initial measured value. The Nominal value can be edited.

Label section

Show check box: When checked, the Caliper label and graphic are displayed in the Graphic Display window.

Show heading check box: Toggles the display of the row and column headings in the Caliper label. When checked, the label's row and column headings are displayed.

Contents area

> **NOTE:** The order you select the following check boxes define the order they appear in the label. The order number appears to the left of each item selected. When you click a marked check box to unmark it, the order numbers of the remaining marked check boxes are reordered accordingly.

Measured check box: When checked, the measured data is displayed in the label.

Nominal check box: When checked, the nominal data is displayed in the label.

Tolerance check box: When checked, the tolerance data is displayed in the label.

Deviation check box: When checked, the deviation data between the measured and nominal values is displayed in the label.

OutTol check box: When checked, the out of tolerance data is displayed in the label.

Default button: Click to set the current selection of check boxes as the default.

Reset button: Click to clear all check box selections in the **Contents** area. The software then resets the section to the auto setting configuration showing the Measured value.

Report section

Show Text Results check box: When this check box is marked, the software outputs the results in text format to the report. The items that appear in the text format output are defined by the Dimension Format command in your measurement routine. For details, see the topic "Dimension Format" in the PC-DMIS Core documentation.

Clear button: Click to reset the **Gage** dialog box to the auto settings configuration.

Create button: Click to create a new Caliper defined with the settings you made in the **Gage** dialog box. The software creates the Caliper.

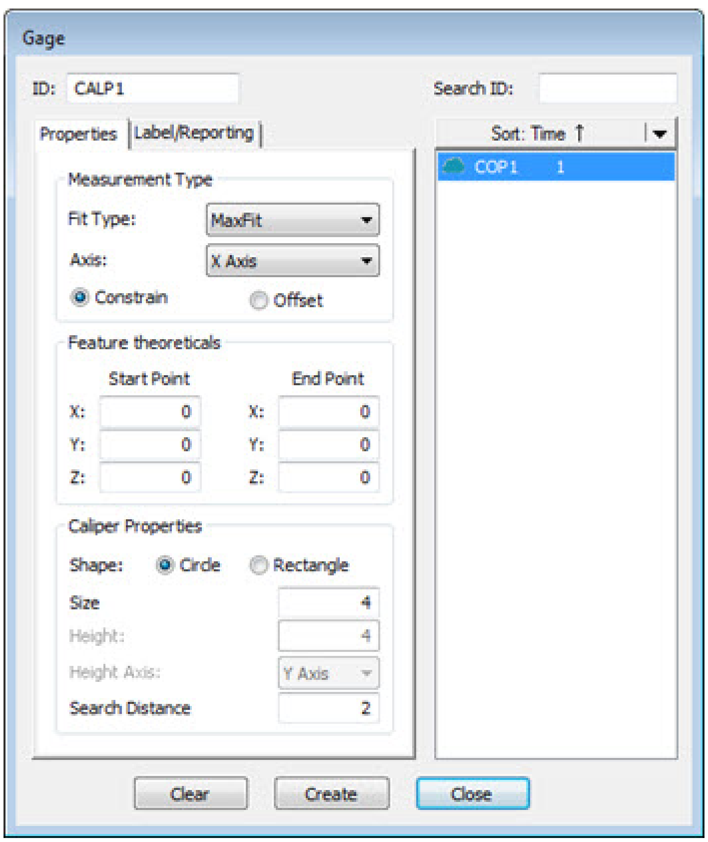
Close button: Click to close the **Gage** dialog box without creating a Caliper.

Creating a Caliper

To create a Caliper feature:

1. Select the **Caliper** option from the **Insert | Gage** menu. The **Gage** dialog box is displayed.

> **HINT:** You can also click the **Caliper** button  from the **QuickCloud** or **QuickMeasure** toolbar to open the **Gage** dialog box.



Gage dialog box

2. Select the COP, COPOPER or Mesh data object to use.
3. From the **Measurement Type** section, select the **Fit Type**.

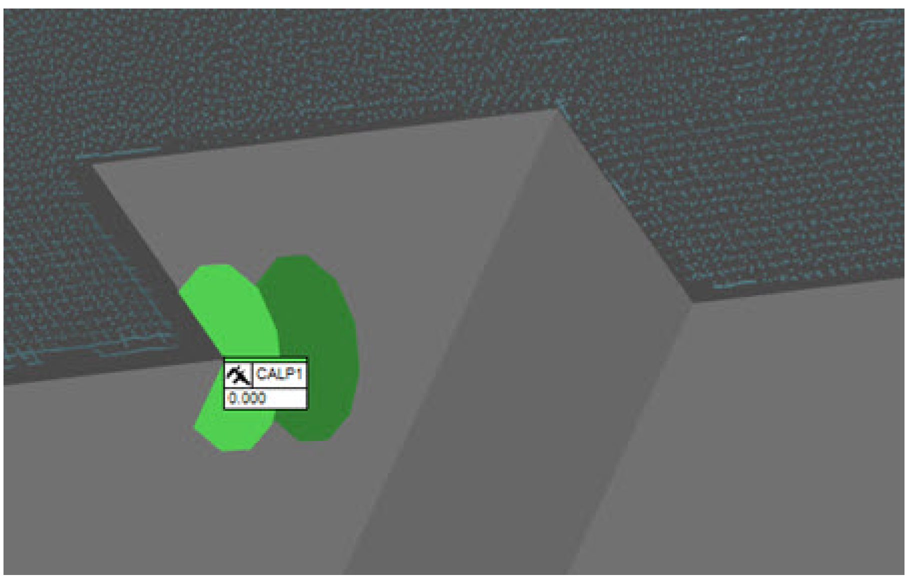
4. Select the **Axis**, and then select the **Constrain** or **Offset** option.
5. From the **Caliper Properties** section, select **Circle** or **Rectangle** from the **Shape** option.
6. Enter the tip **Size** (for Circle tip shape) or **Width** (for Rectangle tip shape), **Height**, **Height Axis**, and **Search Distance**.

The default values are:

- **Size** and **Width** properties: 4 mm
- **Height** property: 4 mm
- **Height Axis**: X Axis
- **Search Distance**: 2 mm

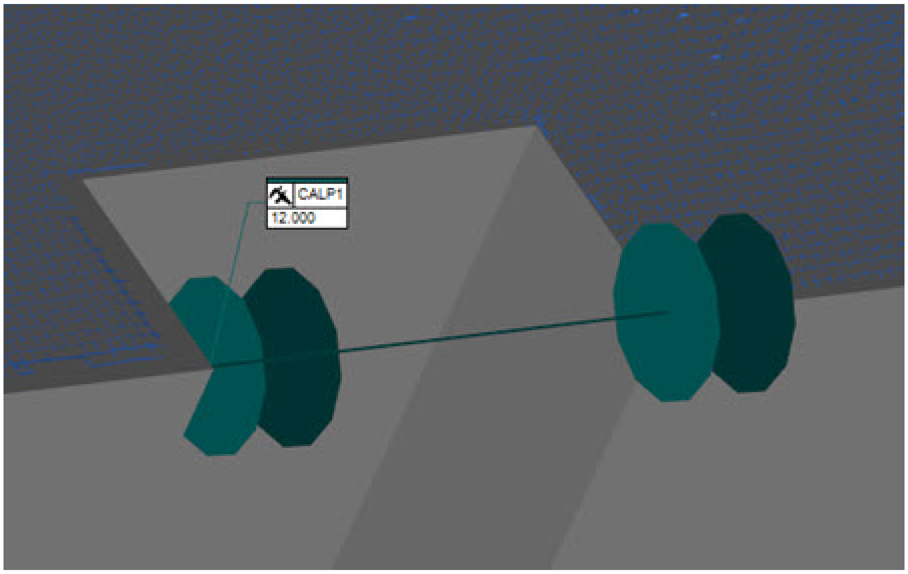
NOTE: Changes to any of the **Gage** dialog box properties become the default values the next time the dialog box is opened.

7. From the Graphic Display window, click to define the start point. To remove the first selected point, press the Delete key on your keyboard.

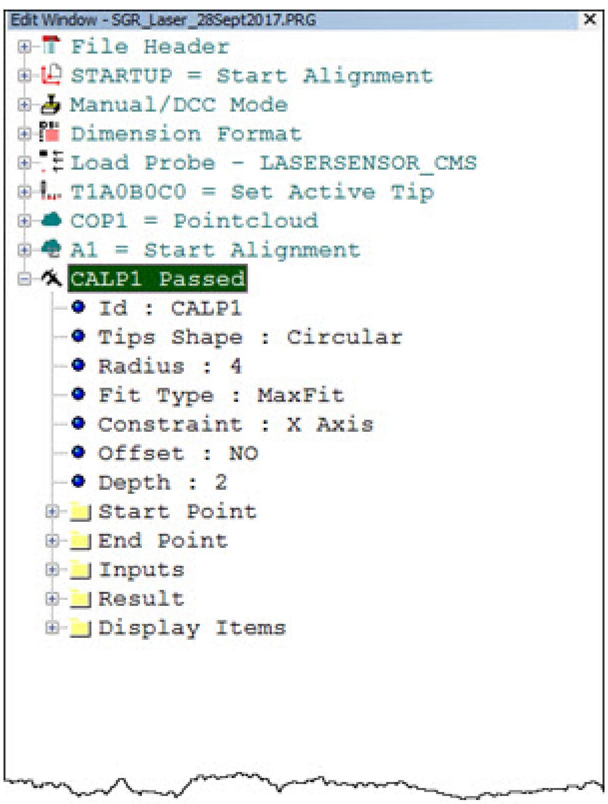


8. Move your cursor to the second location, then click to define the end point. As you move the cursor, the length value updates in the Graphic Display window. If the selected object (COP or Mesh) contains data, the length shown is the measured value. If the selected object is empty and a CAD model is used, the length value shown is the nominal value.

HINT: You can also enter the XYZ values for each in the **Start Point** and **End Point** XYZ boxes.



9. Click **Create** to define the Caliper and add it to the commands in the **Edit** window.



Pointcloud Operators

The Pointcloud operator commands listed below perform different operations on Cloud of Points (COP) commands and other Pointcloud operator commands. The units for these commands are in millimeters.

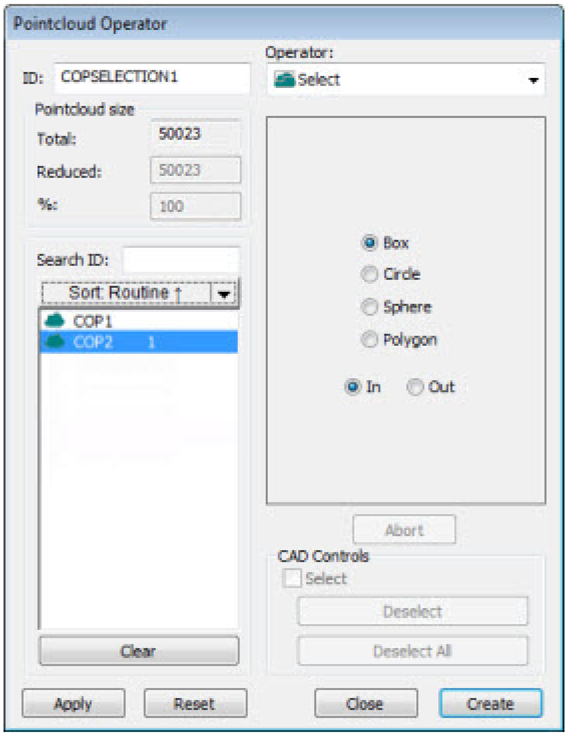
NOTE: Versions earlier than PC-DMIS 2014 used a COPOPER keyword prior to the operator command. This COPOPER command is no longer available and the commands now use a COP prefix. For example, the Filter operator is now COPFILTER.

You can add Pointcloud operator commands into your measurement routine in any of these ways:

- Select the **Insert | Pointcloud | Operator** menu item.
- Select menu items from the following submenus:
  - **File | Import | Pointcloud** - Import from data files to a COP.
  - **File | Export | Pointcloud** - Export to data files from a COP.
  - **Insert | Pointcloud** - Add basic pointcloud commands from this sub-menu. These include COP, and specific point cloud operator commands (**Cross Section**, **Surface Colormap** or **Point Colormap**) that alter the display of Pointclouds in the Graphic Display window.
  - **Operation | Pointcloud** - Alter the number of points that are included in COP commands. Items included in this submenu are: Clean, Empty, Filter, Purge, **Reset**, and Select.
- Manually type Pointcloud operator command into the Edit Window. If the cursor is on the command in the Edit window, and you press **F9**, the **Pointcloud Operator** dialog box opens.
- From the **Pointcloud** toolbar, click the appropriate **Pointcloud Operator** button to open the associated **Pointcloud Operator** dialog box. The Pointcloud Operator is applied to the COP.

IMPORTANT: You must be licensed with the **COP** option to use Pointcloud operator commands. You cannot use these commands if you are only licensed with the Vision option. **Vision** should be disabled when using Laser.

Manipulating Pointcloud Operators



Pointcloud Operator dialog box

The **Pointcloud Operator** dialog box is displayed by selecting **Insert | Pointcloud | Operator** from the main menu. The dialog box contains the following elements:

ID - Contains a unique identity of the pointcloud operator command being edited.

Pointcloud size - This area contains the **Total** size of the pointcloud operator selected in the list box. The **Reduced** size and the percentage (**%**) of reduction in size are also shown.

Command List - The list of commands on the left shows COP or pointcloud operator commands that send data to the pointcloud operator command in the ID box. The Command List section also has these two functions:

Search ID - If there's a long list of operators defined, you can search using the **Search ID** field to locate specific operators in the list. Begin entering the operator's ID into the field and the list will automatically filter based on the entry.

Sort - The **Sort** functionality is available to organize the list by **ID**, **Type**, **Routine** or **Time**. Select the option from the drop-down list then click the **Sort** button.

Apply - Applies the operator to the COP or pointcloud operator commands selected.

Reset - Restores all the data stored in a COP command.

CAD Controls - Lets you apply the operation to selected CAD elements. See "CAD Controls Area" where scanning is discussed for a more detailed description.

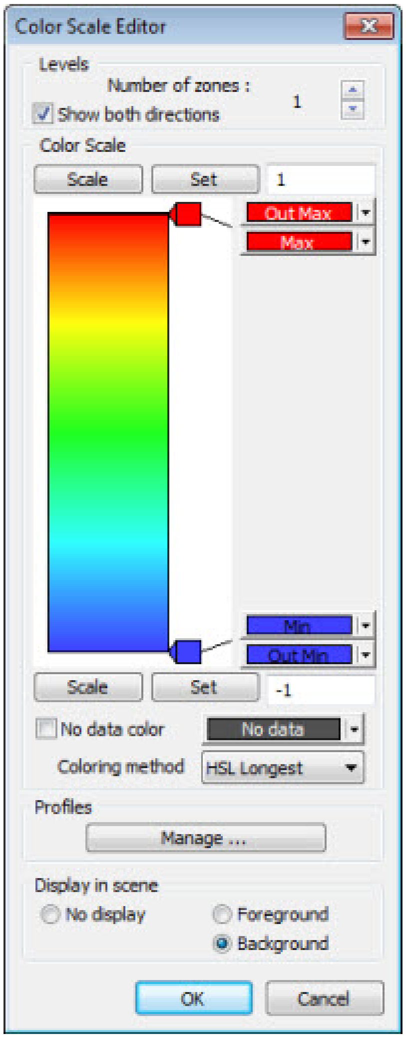
Operator - This list shows the operator commands you can select and apply to pointcloud or other operator commands. Depending on the type of operator selected, different options become available in the dialog box. Refer to the following operator types for details:

Edit the Color Scale

The **Edit Color Scale** button is available on the **Pointcloud Operator** dialogs for the Point Colormap and Surface Colormap operators. It allows you to change the color scale for these operators. By default the Min/Max values of the scale are set to the +/- Tolerance values of the colormap. Different color bars can be saved and recalled using this function.

To begin:

1. With the **Pointcloud Operator** dialog box displayed, select the Surface Colormap or Point Colormap operator.
2. Click the **Edit Color Scale** button on the dialog to display the **Color Scale Editor** dialog:

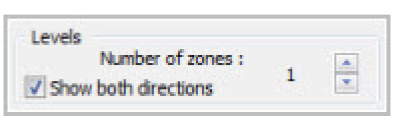


Color Scale Editor dialog box

The following sections of the dialog are described.

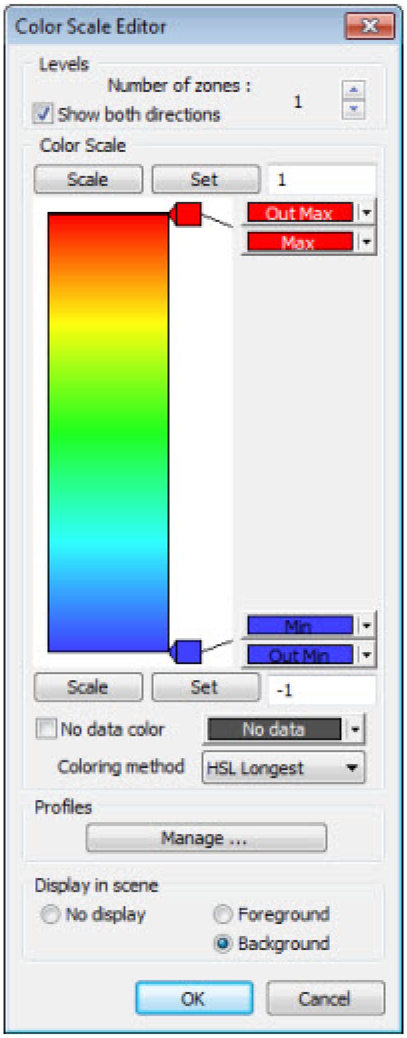
- Levels
- Color Scale
- Profiles
- Display in Scene

Color Bar Levels Area



Levels area of the Color Scale Editor dialog box

Number of zones - Allows you to change the number of color zones displayed in the color bar. A setting of one (1) displays the gradient view as shown below:

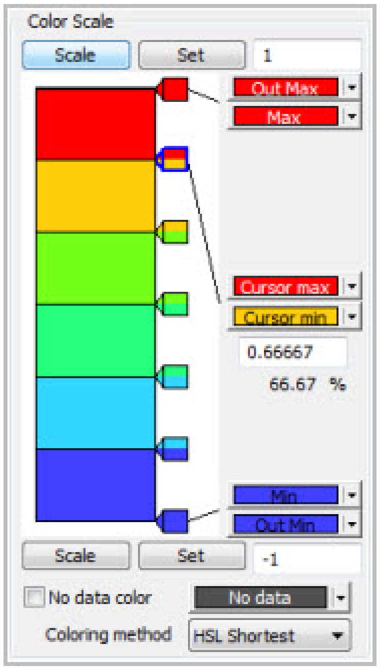


Color Scale Editor dialog box

Change the number of tolerance zones by clicking the up/down **Levels** arrows. You can also just click in any of the current zones to create a new zone at that location.

When the **Show both directions** check box is not marked, the Min value **Scale** and **Set** controls are disabled. The Min value in this case is the negative of the Max value.

Color Bar Color Scale Area



Color Scale area of the Color Scale Editor dialog box

Color Scale section - Determines the tolerance zones and the colors associated with the measured values in relation to the respective tolerances. The **Scale** and **Set** buttons change the Max or Min tolerance values with the following differences:

> **Scale** button - When clicked, the intermediate zone values designated by the tolerance markers are appropriately scaled around the new Max and Min values.
>
> 1. Enter a new Max or Min value then click **Set**. If the Min/Max values on the color bar are changed, this also changes the Plus/Minus tolerance values on the colormap.
> 2. Click the respective **Scale** button. All the zones in the color bar appear the same except the values of each marker are scaled appropriately around the new Max and Min values.
>
> **Set** button - Used to change the upper value of the highest zone or the lower value of the lowest zone. The intermediate zone values designated by the tolerance markers remain the same.
>
> 1. Enter a new Max or Min value.
> 2. Click the respective **Set** button. The corresponding Max or Min zone changes accordingly. All intermediate zone values remain the same.

NOTE: To change Zone values, click and drag one of the zone markers. You can also enter the Zone values. To enter new zone values:

Click the zone marker to display a leader line from the marker to the zone selected and a field appears.

Enter an appropriate value in the field then click outside the field for the value to take affect.

No data color check box - When checked, allows you to select the color where no data exists based on the colormap Max Distance. To define the color for this option:

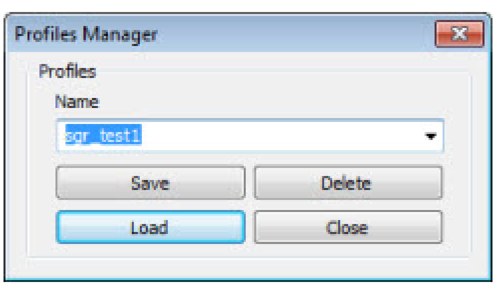
1. Click the drop down arrow to the right of the check box to display the standard color picker dialog.
2. Select the color for this option and click **OK**.
3. Click the check box to mark it and apply this option to your surface colormap.

Coloring method - The drop down list provides pre-defined color bar color schemes you can select. Click the drop-down arrow to display the list and select the option you want to apply.

Color Bar Profiles Area

The **Profiles** area of the **Color Scale Editor** dialog is used to manage color bar schemes.

Click the **Manage** button to display the **Profile Manager** dialog box.



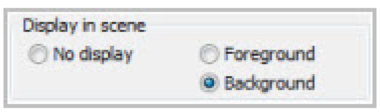
Profile Manager dialog box

The following options are available:

- If this is a new color scheme, enter a unique name for the color scheme in the **Name** field and click **Save**. The current color bar profile is saved under the name entered.
- Load a profile by selecting one from the drop down **Name** list and click **Load**. You can also begin typing a name in the **Name** field to filter the list based on your entry.
- Delete an existing profile by selecting one from the drop down **Name** list and click **Delete**. You can also begin typing a name in the **Name** field to filter the list based on your entry. The selected profile is permanently deleted - this cannot be undone so use caution when deleting color schemes.

NOTE: The files are saved as .cbr files in the same folder as the measurement routines are saved.

Color Bar Display in Scene Area



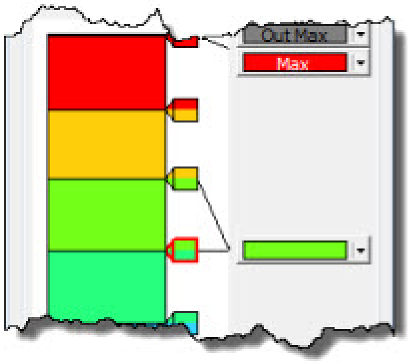
Display in Scene area of the Color Scale Editor dialog box

The Display in Scene area of the Color Scale Editor dialog box is used to define how the color scheme is to be presented in the Graphic Display window. The options are:

- No Display - The Color Bar is not displayed in the Graphic Display window.
- Foreground - The Color Bar is displayed on top of the CAD objects in the Graphic Display window.
- Background - The Color Bar is displayed behind the CAD objects in the Graphic Display window.

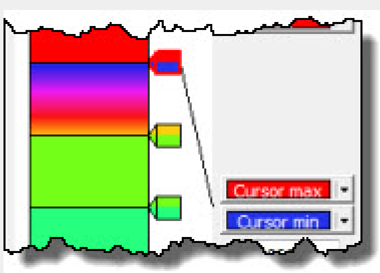
Changing a Zone's Color

1. Click the Max tolerance marker  for the particular zone then press the Ctrl key on your keyboard and click the Min tolerance marker for the same zone.

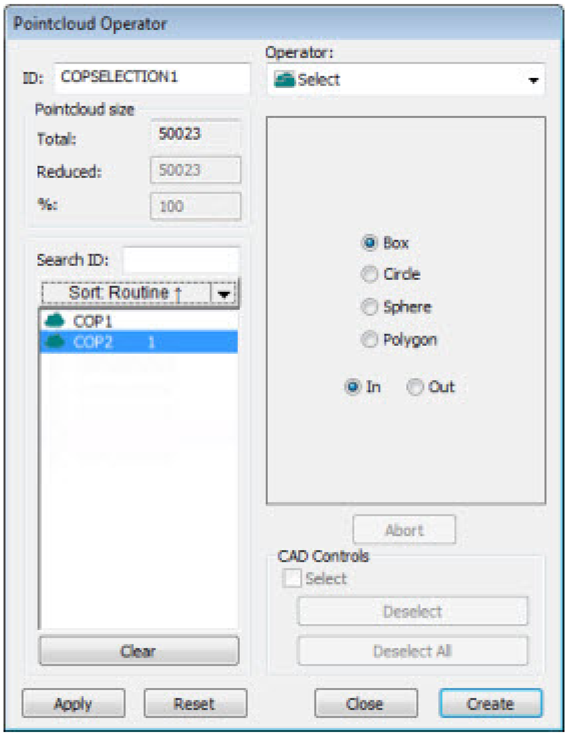


2. Once selected, click the drop-down arrow to display the standard color-picker dialog.
3. Select the new color and click **OK**. The color for the selected zone is changed to the new color.

EXAMPLE: Changing only the Max or Min value of a zone only changes that zone's color to a gradient scheme. For example, if you change just the Max color of a zone, the zone's gradient color scheme is based on the new Max color selected and the current color of the Min value as shown below.



SELECT



Pointcloud Operator dialog box - SELECT Operator

The SELECT operation selects a subset of data contained in a COP command.

 To apply the SELECT operation to a Pointcloud, click **Select Pointcloud** from the **Pointcloud** toolbar, or select **Operation | Pointcloud | Select**. By default, the **Polygon** option is used when you select the **Select Pointcloud** button the toolbar.

To select a region of points:

1. Select the desired option button inside the dialog box:

   Box

   Circle

   Sphere

   Polygon

   > **HINT:** Press the **End** key to close the polygon selection.

2. Select the **Pointcloud** command that you want to apply the selection to form the list of commands.

3. Make the selections that define your selection type by clicking and dragging in the CAD in the Graphic Display window. The axis of the selection entities should be perpendicular to the current view. See the table below as a guide for what you should select.
4. If you want to keep the points inside the selection domain, select **In**. If you want to keep the points outside the selection domain, select **Out**.
5. After clicking the necessary points in the Graphic Display window, click the **Apply** button to define the selection type. PC-DMIS displays the points inside or outside the selected domain in the Graphic Display window. If you use the **Sphere** selection type, the closest pointcloud point is used for the center of the sphere.
6. When you're finished, click **Create**. PC-DMIS inserts a `COP/OPER,SELECT` command.

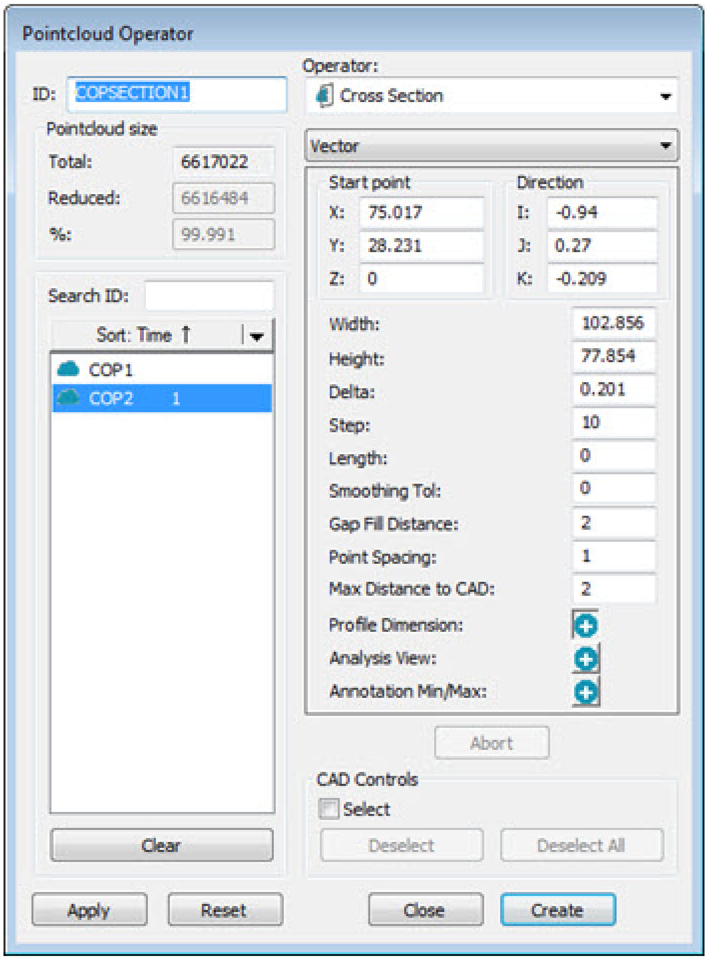
> **NOTE:** If you want to select the complement data instead, you can use the BOOLEAN operator to do that. For information on the **Complement** option inside BOOLEAN, see the "BOOLEAN" topic.

Type	Points Needed
Box	Select two corners.
Circle	Select the center and a point specifying the radius of the circle.
Sphere	Click one point. PC-DMIS projects it onto the cloud of points to find the closest point. This represents the center of the selected sphere. Click another point. PC-DMIS uses this second point to determine the radius of the sphere.
Polygon	Select the vertices of the polygon. Press the End key to close the polygon.

Click **Create** inserts a `COP/OPER,SELECT` command into the Edit window.

> **EXAMPLE:** For example:
>
> `COPSELECT4=COP/OPER,SELECT,BOX,SIZE=27377`
>
> `REF,COP1,,`

CROSS SECTION



Pointcloud Operator dialog box - CROSS SECTION Operator

The CROSS SECTION operation generates a subset of polylines determined by the defined intersection of a set of parallel planes with the COP or Mesh object. The set of planes is defined by the start point, the direction vector, the step distance between the planes, and the length. The number of planes is determined by the **Step** distance divided into the **Length** plus one.

NOTE: The CROSS SECTION operator can be evaluated by the profile dimension.

To apply the CROSS SECTION operation to a Pointcloud, click **Cross Section Pointcloud** on the **Pointcloud** toolbar, or select **Insert | Pointcloud | Cross Section**.

From the **Pointcloud** or **QuickCloud** toolbar, click the **Cross Section Slide Show** button to display cross sections in 2D view. For details, see the "Cross Section Slide Show" section of the "Show and Hide Cross Section Polylines" topic.

The list underneath the **Operator** list contains these options: **Vector**, **Axis**, **Curve**, and **2 Points**. For details on how the **Curve** function works, see the "Creating a Cross Section along a Curve" topic. For details on the **2 Points** option, see the "Creating a Cross Section between 2 Points" topic.

The CROSS SECTION operator uses the following options:

- **Start Point**: Indicates the coordinates of a point belonging to the first plane cutting the pointcloud. It is displayed in the Graphic Display window as a blue ball that is used as a handle to drag to a new location when needed. It is defined by the first click in the Graphic Display window. In the actual Edit window command, the start point value is held in the START PT parameter.
- **Direction** (applies only to the **Vector** and **2 Points** option): This value indicates the direction of the normal vector. It can be defined by the first click in the Graphic Display window. In the actual Edit window command, the **Direction** value is held in the NORMAL parameter.
- **Axis** (applies only to the **Axis** option): Use this option to create a cross section along the X, Y, or Z axis. Select the desired axis (the default is X), set a start point in the Graphic Display window, and set an end point. The section plane will cut the part at a given step value over the length of the cross section.
- **Width**: This value indicates the width of the section under consideration. If the value is 0, the system calculates the value as the CAD and COP bounding box value.
- **Height**: This value indicates the height of the section under consideration. If the value is 0, the system calculates the value as the CAD and COP bounding box value.
- **Delta**: This value indicates the maximum distance from the plane for a point to be considered part of the cross section. In the actual Edit window command, the **Delta** value is held in the TOLERANCE parameter. The **Delta** property is only available when a COP object is selected.
- **Step**: This value indicates the distance between the planes. In the actual Edit window command, the step value is held in the INCREMENT parameter.

> **HINT:** If the **Step** value is greater than the Length value, then only one section cut is created at the start point.

- **Length**: This value indicates the maximum distance between the first and last plane. The length value is displayed in the **Length** parameter of the dialog box, and shown as a purple line in the Graphic Display window.
- **Smoothing Tol**: Set to 0 (zero) to turn off smoothing (default value).

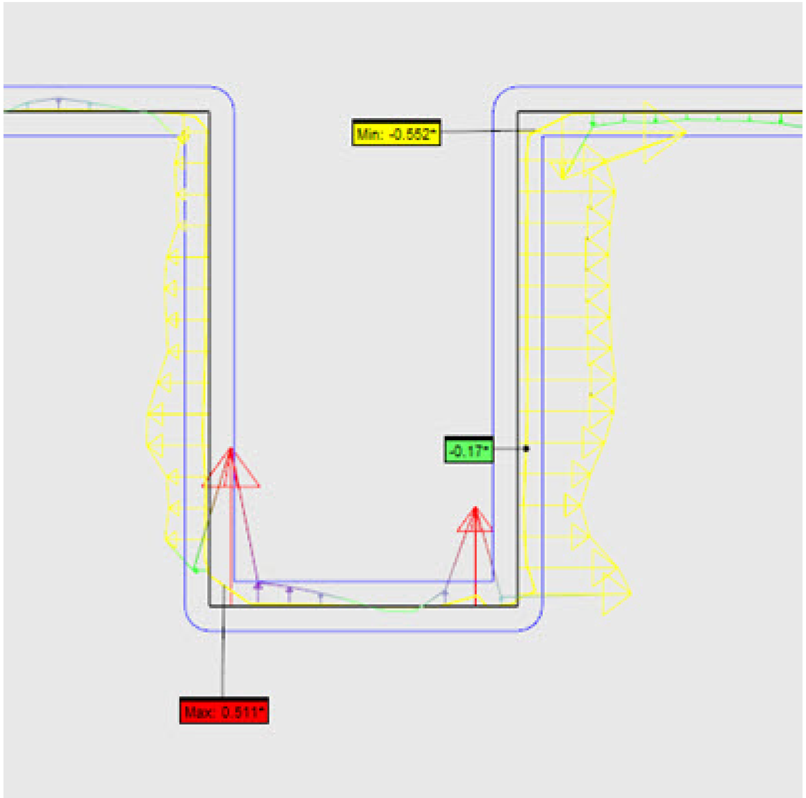
Use **Smoothing Tol** to remove small steps in the cross section and create a smoother measured polyline. This setting filters out the points within the smoothing tolerance value and then fits a polyline to the data using the **Point Spacing** value.

HINT: The **Point Spacing** is also defined by the `CrossSectionCopCadCrossSectionStep` registry setting. For details on this registry setting, see the "`CrossSectionCopCadCrossSectionStep`" topic found in the Settings Editor documentation.

The **Smoothing Tol** should be set very small so that the measured cross section does not deviate greatly from the actual data. Except for extreme situations (for example, a very large CAD model and / or very poor density of points), this parameter should be set between a few tenths of a mm (maximum) and a few thousandths of a mm (minimum).

- **Gap Fill Distance**: Defines the maximum gap distance along the yellow measured polylines of a cross section. If gaps equal to or smaller than this value appear, the gaps are filled in with calculated points. This value can also be set in the Settings Editor. For details, see the "`CrossSectionMaximumEmptyLength`" topic in the Settings Editor help.
- **Point Spacing**: This entry is used only when the `CrossSectionCopCadCrossSectionDrivenByCad` registry entry is set to 1 (True). This value is the step used along the CAD polylines to look for the best interpolated COP point. For greater accuracy, or if the CAD model is very small, this value can be set to a smaller value. This value can also be set in the Settings Editor. For details, see the "`CrossSectionCopCadCrossSectionStep`" topic in the Settings Editor help.
- **Max Distance to CAD**: The value entered defines the maximum distance of the pointcloud data relative to the nominal CAD model. The default value is 2 mm. If the scanned part deviates more than the Max Distance value from the CAD model, the software may not compute the yellow measured cross section. You can adjust this value to account for large deviations of the scanned data relative to the CAD model.
- **Profile Dimension**: Click the **Add** button  to create a new profile dimension for each cross section. For details on Profile Dimension, see the "Dimensioning Profile - Line or Surface" chapter of the PC-DMIS Core help.
- **Analysis View**: Click the **Add** button to create the `ANALYSISVIEW` command in the Edit window. For details on the `ANALYSISVIEW` command, see the topic "Create Analysis View Command" in the PC-DMIS Core help.

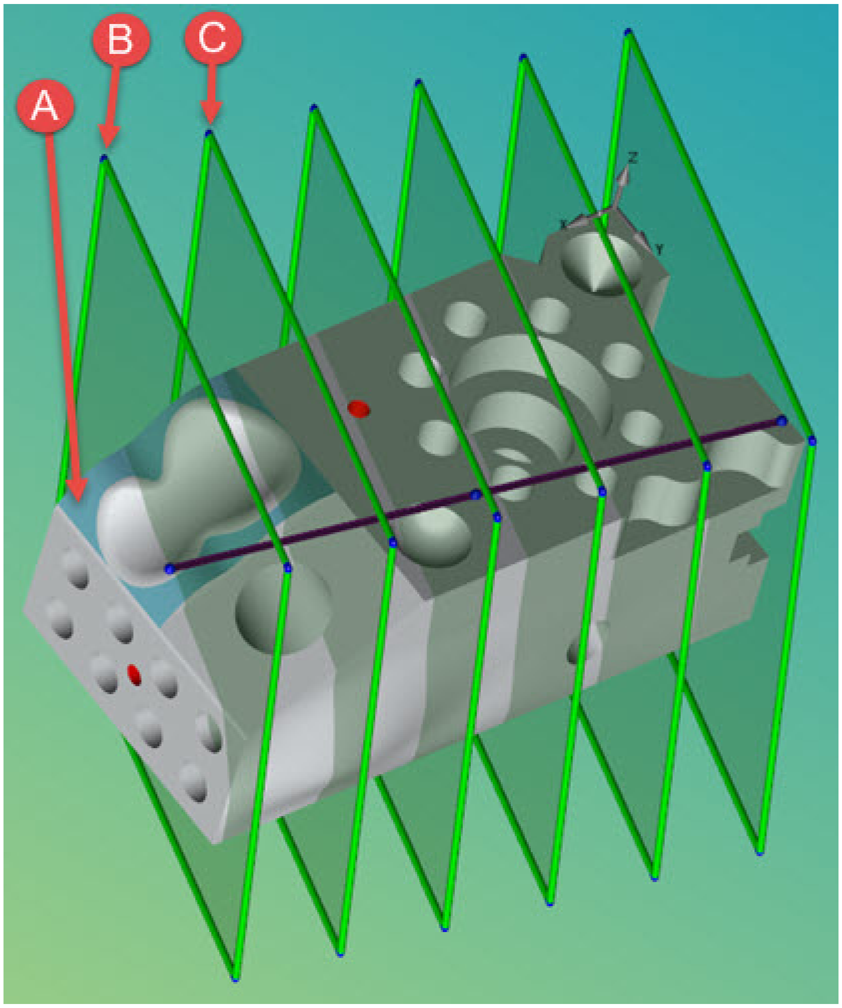
- **Annotation Min/Max**: Click the **Add** button to create the minimum and maximum values in the form of annotation labels for the active cross section.



The minimum and maximum points are re-calculated each time the measurement routine is executed.

- **CAD Controls**: Mark the **Select** check box to select surfaces in the Graphic Display window. PC-DMIS filters out any cross sections that do not pass through the selected surfaces when you click **Create**.

For example, if you selected surface A after the start and end points are defined, only the cross sections at B and C would be generated:



Example of a Selected Surface (A) limiting the cross sections to only (B) and (C)

Selected surfaces do not affect what you see when clicking the **View** button.

When the cutting planes are visible in the Graphic Display window, it is possible to manipulate them as follows:

- Select a plane's edge handle and drag to resize the height and width of the cutting planes.
- Select a plane's corner handle and drag to rotate the set of planes around their axis.
- Select the first or last purple length line's blue point handle and drag to redefine the purple line's `START` or `END` definition. While the direction is changing, the values in the dialog box and the number of planes in the Graphic Display window are updated. In the case of Axis mode, the direction of the planes does not change.

- Select and drag the purple length line's middle blue point handle to move the set of planes.

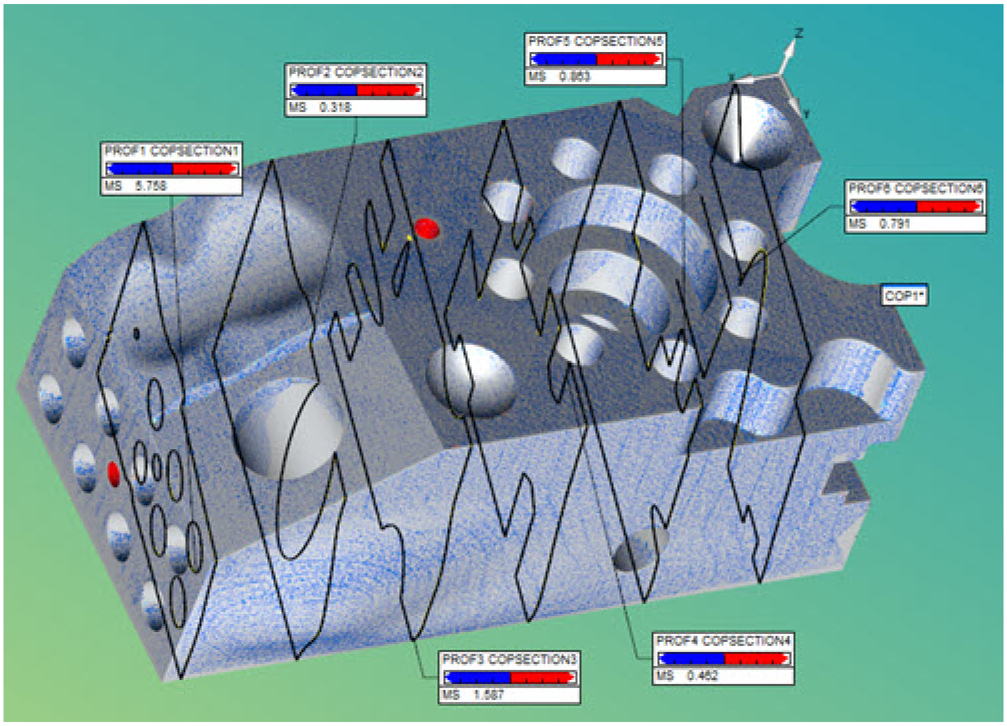
> **NOTE:** When a Cross Section is created or edited, the cutting planes are displayed in a transparent view as shown above.

Click **Create** to:

- Inserts a `COP/OPER,CROSS SECTION` command for each plane into the Edit window.

> **EXAMPLE:** For example:
>
> ```
> COPSECTION3=COP/OPER,Cross
> Section,TOLERANCE=0.05,WIDTH=117.715,HEIGHT=227.086,
>
> START PT = -6.439,60.097,6.276,NORMAL = 0.9684394,-
> 0.2221293,-0.1130655,SIZE=76
>
> REF,COP1,,
> ```
>
> The black polylines represent the nominal CAD, the yellow polylines represent the COP polyline.

- Inserts a label for each plane into the Graphic Display window as shown below:



Finished Cross Sections Showing Six Planes

Defining the Cross Section by Typing Values

Use the **Pointcloud Operator** dialog box to type in the needed values:

- **START PT**: specify the cross section's starting point using **Start Point X**, **Y** and **Z** boxes.
- **NORMAL**: specify the cross section's vector using the **Direction I**, **J** and **K** boxes.
- **WIDTH**: specify the value of the cross section's width property in the **Width** box.
- **HEIGHT**: specify the value of the cross section's height property in the **Height** box.
- **TOLERANCE**: specify the value used to determine the maximum distance from the plane for a point to be considered part of the cross section in the **Delta** box.
- **INCREMENT**: specify the value between cutting planes in the **Step** box.
- **LENGTH**: specify the value between the first and last cutting planes in the **Length** box.
- **SMOOTHING TOLERANCE**: specify the tolerance value to refine the points associated with the generated cross section in the **Smoothing Tol** box.

Defining the Cross Section by Using the Graphic Display Window

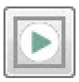
To define some of the cross section parameters, click the CAD model in the Graphic Display window to select the **Start Point**. A pink line appears. Click a second point on the CAD model to determine the **Direction** vector and the **Length**.

Creating a Profile Dimension from the Graphic Display Window

When you double-click a cross section label, a new profile dimension is created that evaluates the selected cross section.

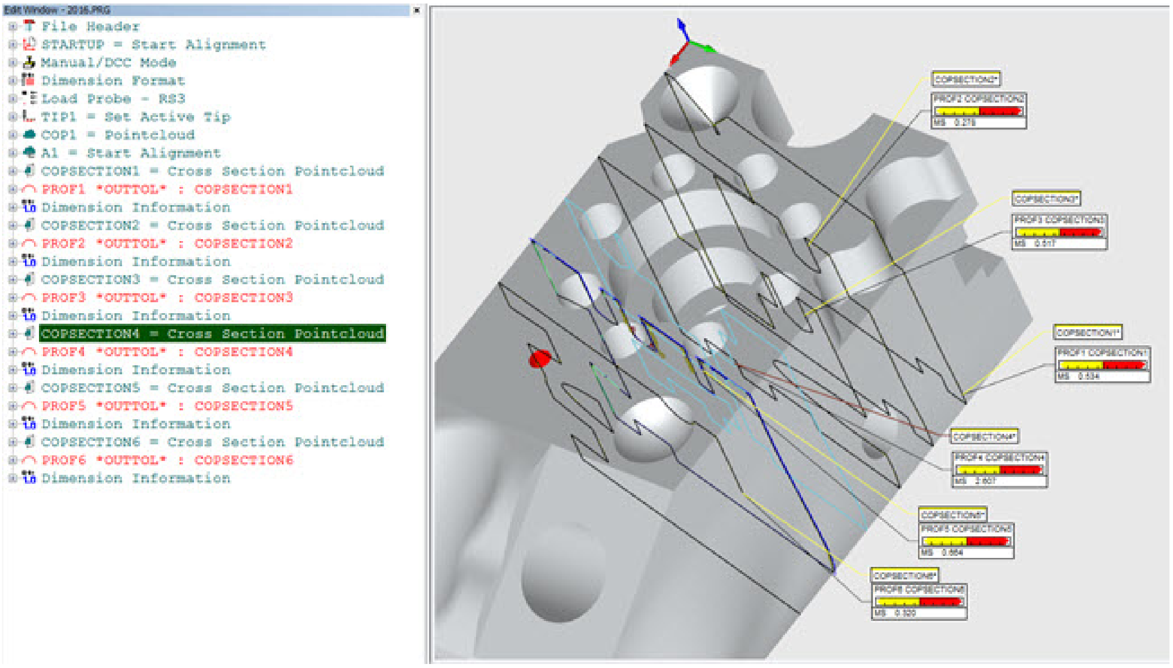
2D View of Cross Sections

Once you have a Cross Section defined, each section can be individually displayed in a 2D view. The view would be normal to the cross section. Any annotation points created on the cross-section appear in the 2D view.

From the **Pointcloud** or **QuickCloud** toolbar, click the **Cross Section Slide Show** button  to display cross sections in 2D view. For details, see the "Cross Section Slide Show" section of the "Show and Hide Cross Section Polylines" topic.

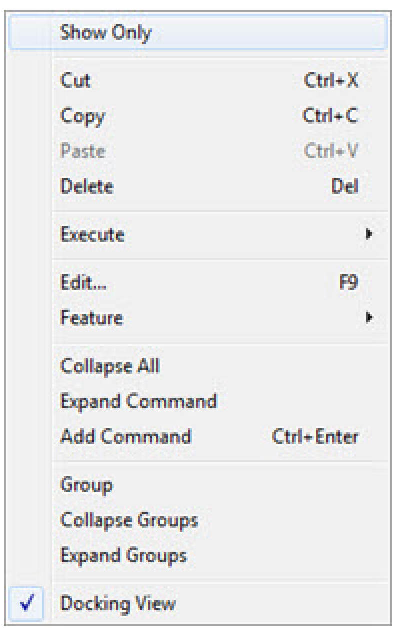
You can also follow these steps:

1. From the **Edit Window**, click the cross section you would like to view to select it. The selected section appears in light blue in the Graphic Display.

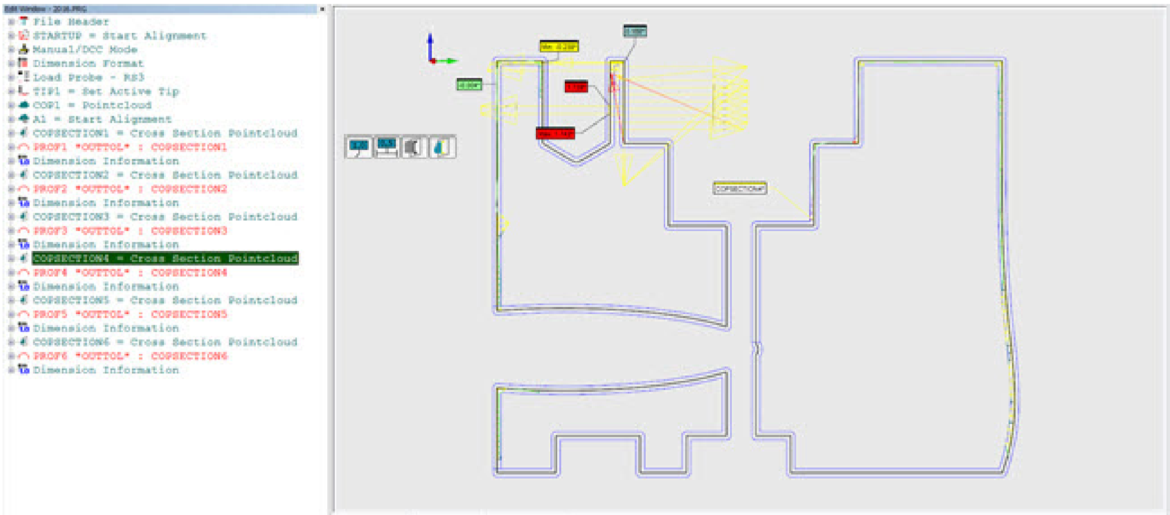


Example of a selected section of a Cross Section

2. Right-click the selected section to display the **Edit Window** pop-up menu.



3. Click the **Show Only** option to display the 2D view of the selected section. A check mark is displayed to the left of the option when enabled.



Example of a section view normal to the Cross Section

> **NOTE:** As you hover and move the cursor over the cross section in the Graphic Display window, the labels are displayed and updated in real time. Click any point on the cross section when in 2D view to create an annotation label for that location.

4. When in 2D view, the **Cross Section Graphic Control** toolbar is available. It is a floating toolbar so it can be positioned anywhere in the Graphic Display window.

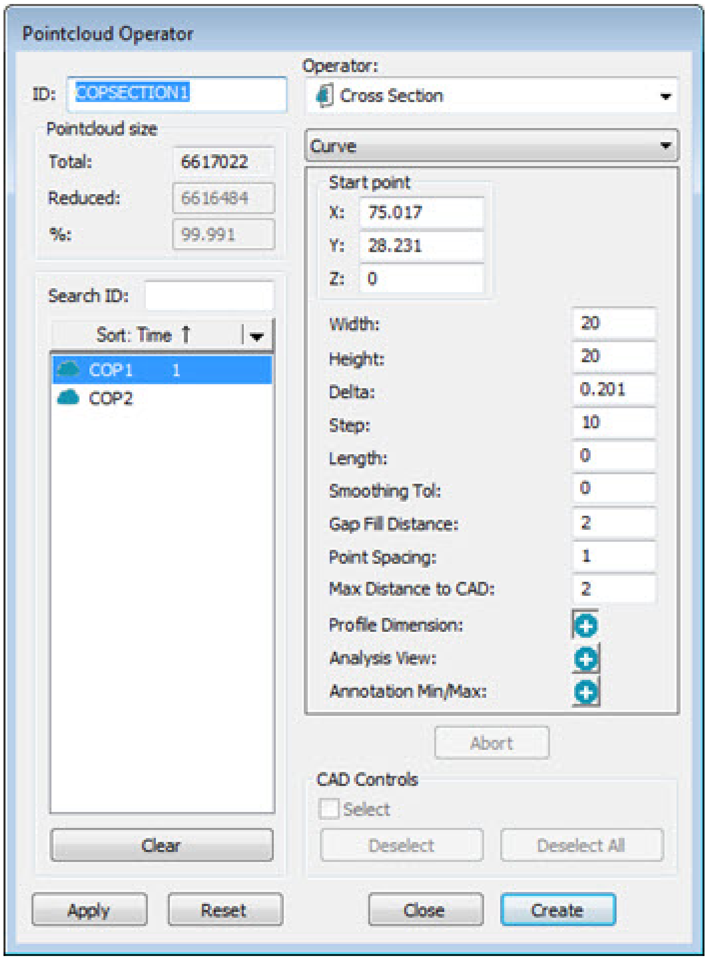
The buttons from left to right perform these functions:

- Show/Hide Annotations
- Show/Hide Distance Gages
- Show/Hide Nominal Polylines
- Show/Hide Measured Polylines

5. Repeat for any other section you would like to view in 2D.

Creating a Cross Section along a Curve

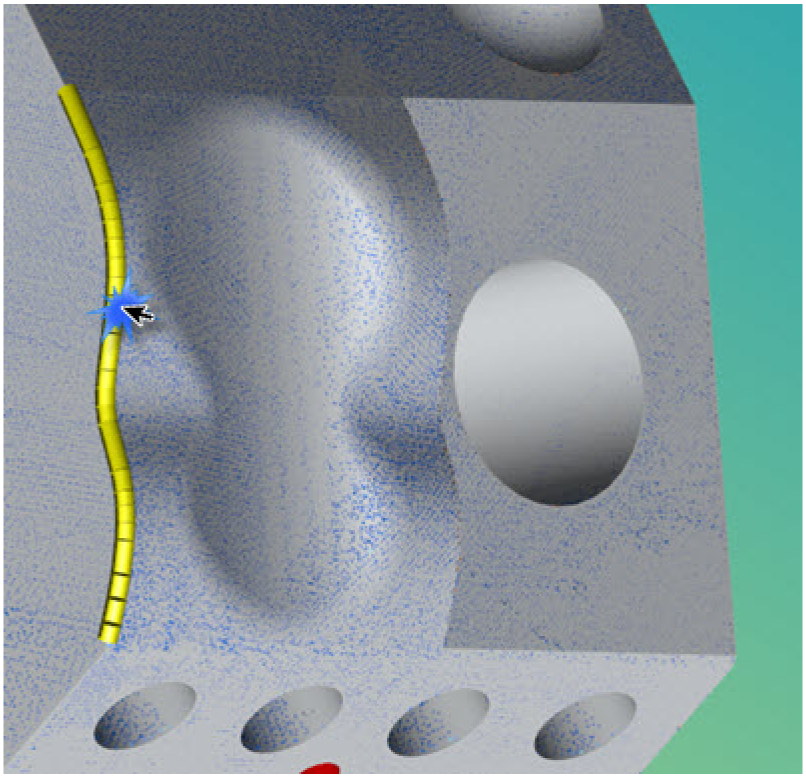
You can create a cross section along a curved feature with the **Curve** function of the **Pointcloud Operator** dialog box. The cross section is created normal to the CAD curve.



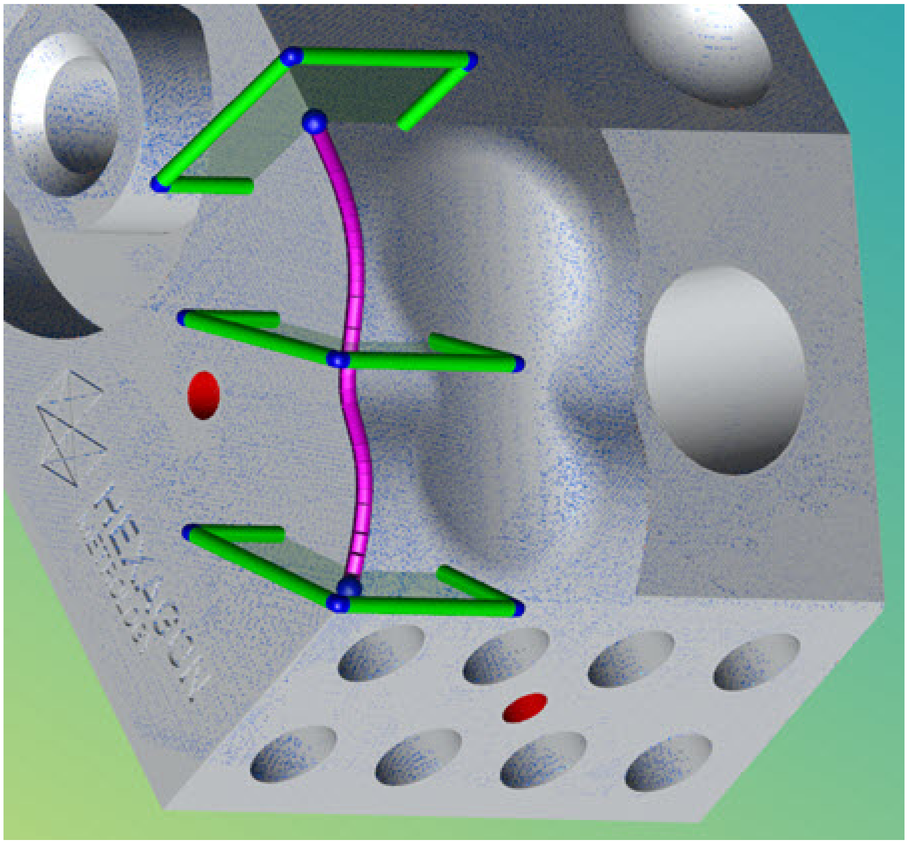
Pointcloud Operator dialog box - CROSS SECTION Operator, Curve function selected

To create a cross section along a curve:

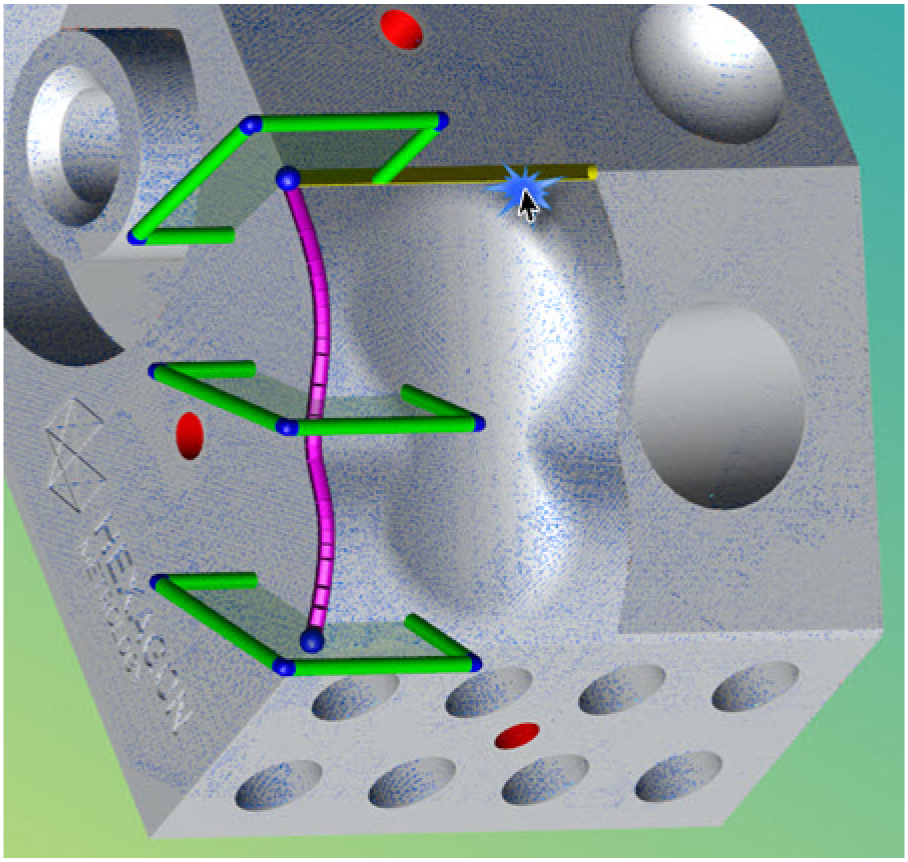
1. Click **Insert | Pointcloud | Operator** to display the **Pointcloud Operator** dialog box.
2. Select the **Cross Section** operator from the **Operator** list, and then the **Curve** function from the list underneath the **Operator** list.
3. In the Graphic Display window, hover over any curved feature and PC-DMIS automatically detects and highlights the curve.



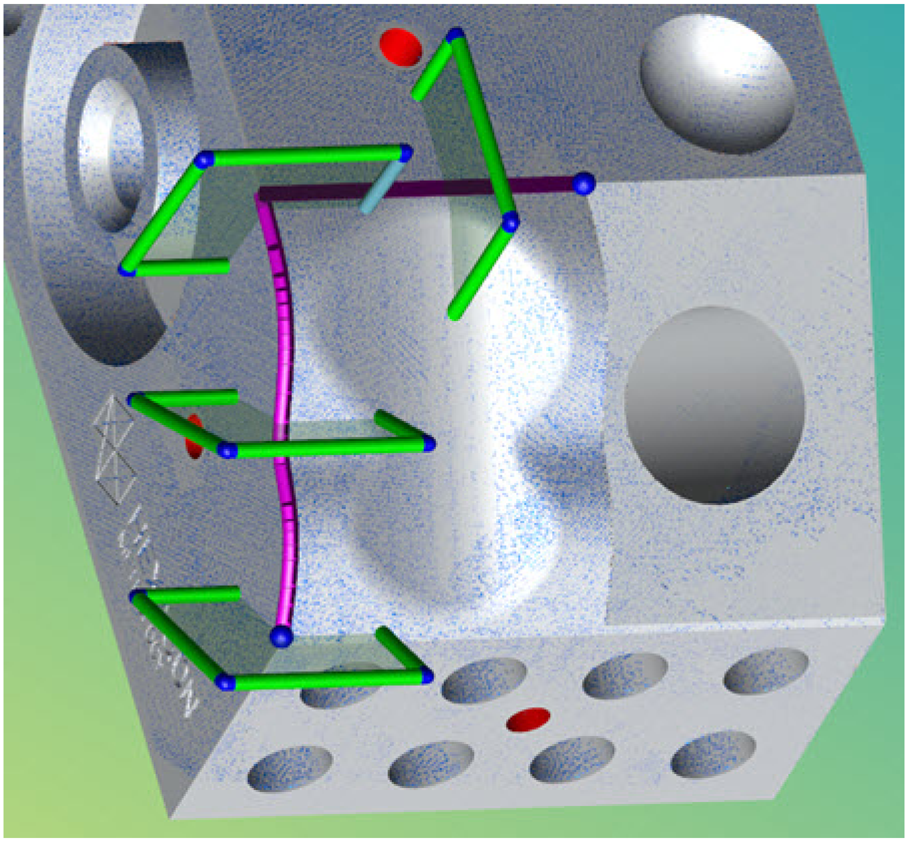
4. Click the highlighted edge you want to create cross sections on. PC-DMIS automatically generates the cross sections.



Hold down the Ctrl key while hovering over the next edge to select multiple contiguous edges.



Click the edge to select it.

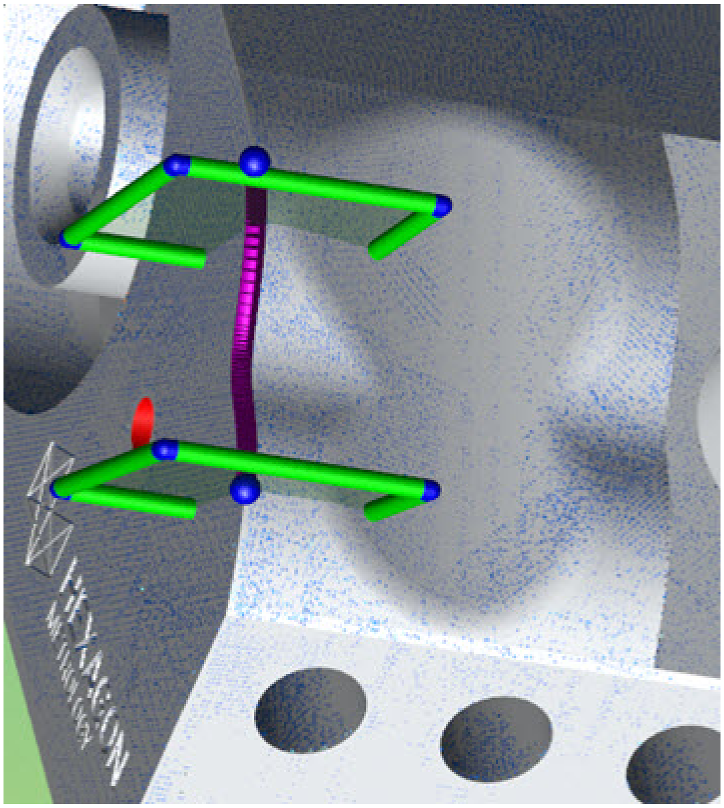


Select as many edges in this manner as required.

To de-select an edge, press the Ctrl key and hover over the first or last edge (it turns red) and then left-click it.

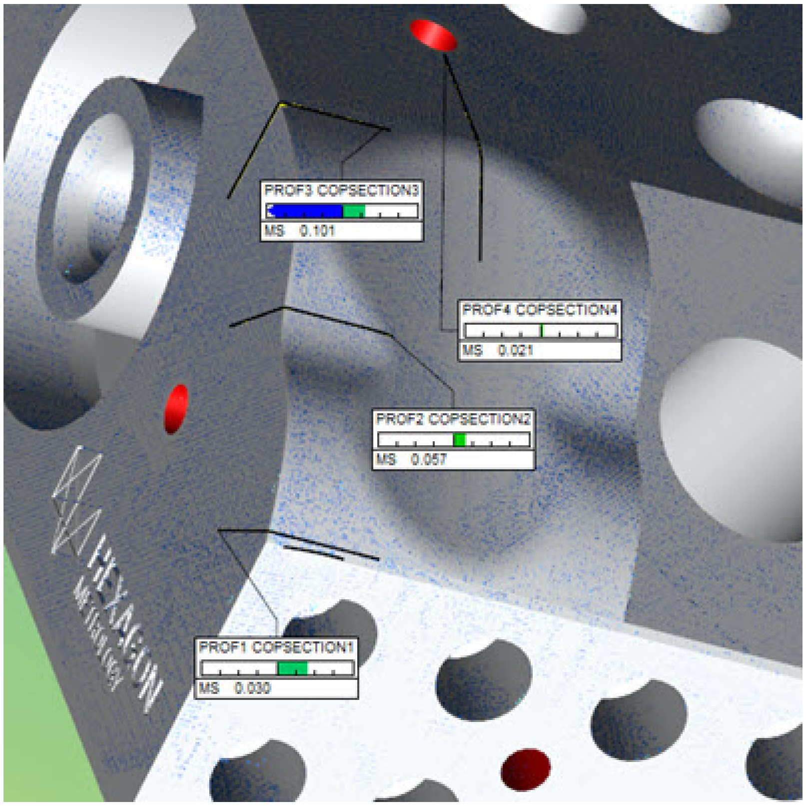
To de-select all edges, click the **Reset** button.

5. Drag the **Start** or **End** points (the blue ball handles) of the curve length line (the purple line) to define only a portion of the curve. If the updated section is too short, click the **Reset** button to cancel and repeat from step 3.



The dialog box values update automatically when changes are made to the **Start** or **End** points of the defined cross section.

6. When done, click **Apply** to create the polylines. Click **Create** to generate the cross sections in the **Edit** window.



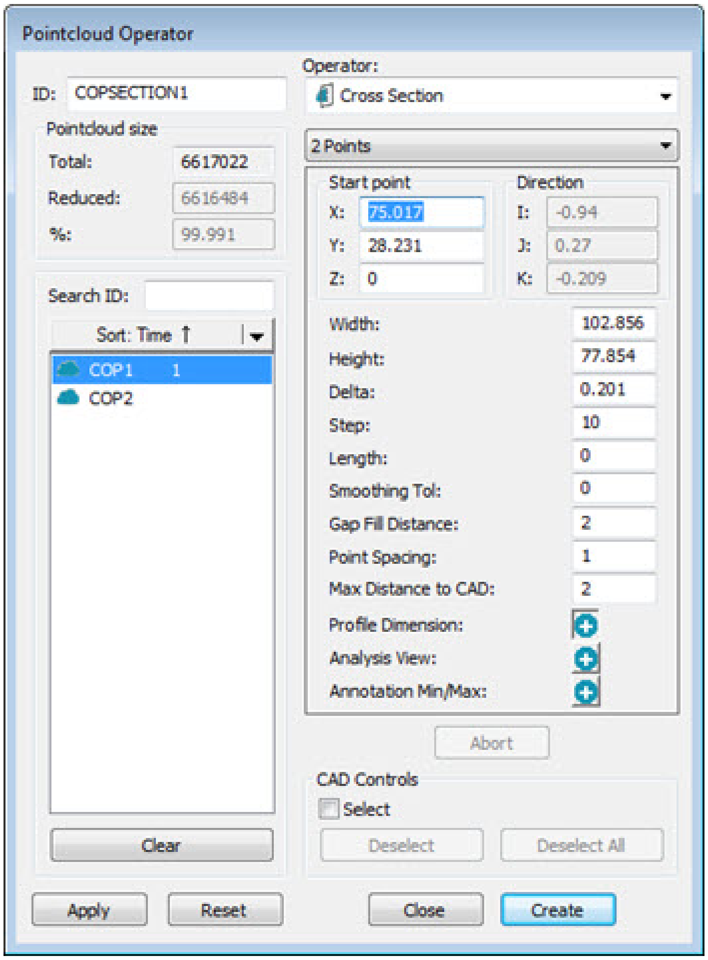
The black polylines represent the nominal CAD, the yellow polylines represent the COP polyline.

Smoothing the cross section along the curve

You can smooth the cross section created along a curve with the Smoothing Tol option in the **Pointcloud Operator** dialog box. For details, see the "Smoothing Tol" description in the "Cross Section" Laser help topic.

Creating a Cross Section between 2 Points

You can create a cross section between two points with the **2 Points** function of the **Pointcloud Operator** dialog box.



Pointcloud Operator dialog box - CROSS SECTION Operator, 2 Points function selected

The 2 Points cross section is:

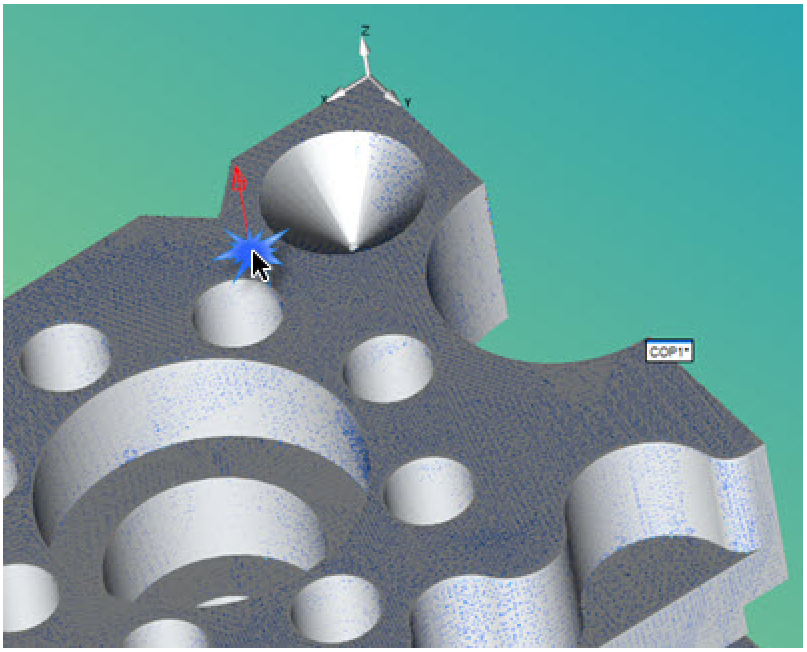
- Created between two selected points and is oriented normal to the current Graphic View
- The cross section's purple Length line is perpendicular to the line defined by the two selected points, it is created at the midpoint of this line and defaults to 0 (zero).

To create a cross section between two points:

1. Click **Insert | Pointcloud | Operator** to display the **Pointcloud Operator** dialog box.
2. Select the **Cross Section** operator from the **Operator** list and then the **2 Points** function from the list underneath the **Operator** list.
3. From the QuickMeasure or Graphic View toolbar, select the correct Graphic View for the cross section orientation. For details on the **QuickMeasure** toolbar, see

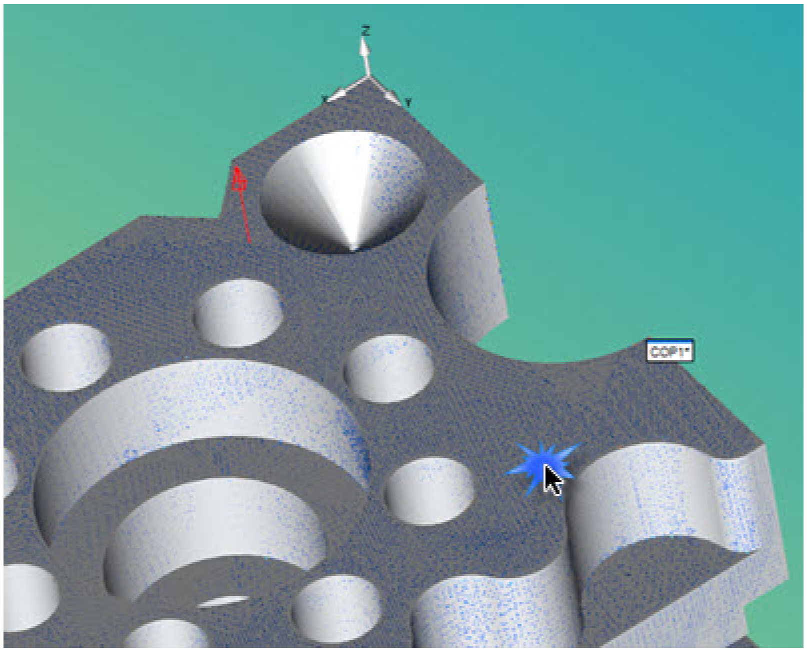
the "QuickMeasure Toolbar" in the "PC-DMIS CMM" documentation. For details on the **Graphic View** toolbar, see the "Graphic View Toolbar" topic in the "Using Toolbars" section of the PC-DMIS Core documentation.

4. From the Graphic Display window, click where you want to define the cross sections's first point.

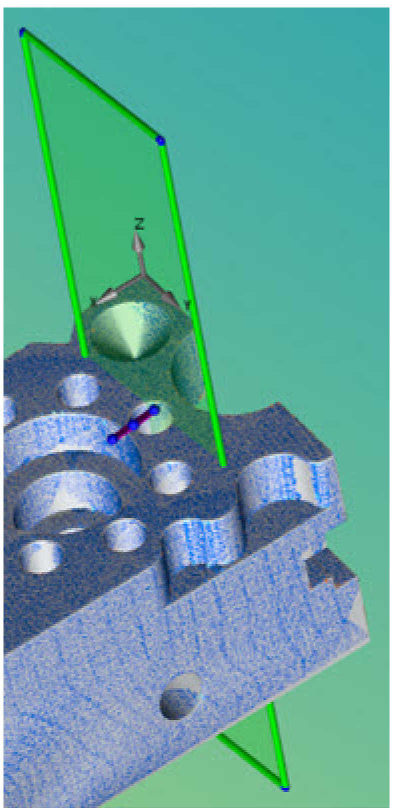


The point's vector is displayed as a red arrow normal to the selected surface.

5. From the Graphic Display window, click where you want to define the cross sections's second point.



Once the second point is clicked, the cross section is displayed.



6. Adjust the cross section's properties as needed.

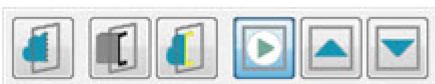
Show and Hide Cross Section Polylines

You can show or hide cross section features that have been created.

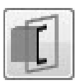
Show or Hide Cross Section Polylines from the QuickCloud Toolbar

1. Display the **QuickCloud** toolbar if not already visible (**View | Toolbars | QuickCloud**).

2. Click the **Cross Section** drop-down arrow to display the **Cross Section** toolbar.



3. Click the appropriate button to perform the described action:



Show All Nominal Cross Section Polylines button. If any of the black nominal polylines are visible, they become hidden. If hidden, they become visible.

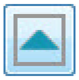
Show All Measured Cross Section Polylines button. If any of the yellow measured polylines are visible, they become hidden. If hidden, they become visible.

Cross Section Slide Show

The **Cross Section Slide Show** button  enables the **Show the Previous Cross Section** and **Show the Next Cross Section** buttons. You can tell that the cross section slide show is enabled when the button appears to be pressed in .

Once enabled, click **Show Previous Cross Section** and **Show Next Cross Section** to display individual cross sections in 2D view (Show Only view) as described below.

1. From the **QuickCloud** toolbar, click the **Cross Section** drop-down arrow to display the **Cross Section** toolbar.
2. Click the **Cross Section Slide Show** button to enable these buttons:



Show Previous Cross Section - Click to display the cross section *before* the currently selected one in the Edit Window in 2D view. The CAD graphic disappears. Click the button repeatedly to cycle backwards until you reach the first cross section.

HINT: If a cross section is not selected, the first one above the current cursor position in the Edit Window is selected. Consequently, nothing happens if there are no cross sections defined above the current cursor position. The same occurs if the *first* cross section in the list is selected and this button is clicked.

Show Next Cross Section - Click to display the cross section *after* the currently selected one in the Edit Window in 2D view. The CAD graphic disappears. Click the button repeatedly to cycle forward until you reach the last cross section.

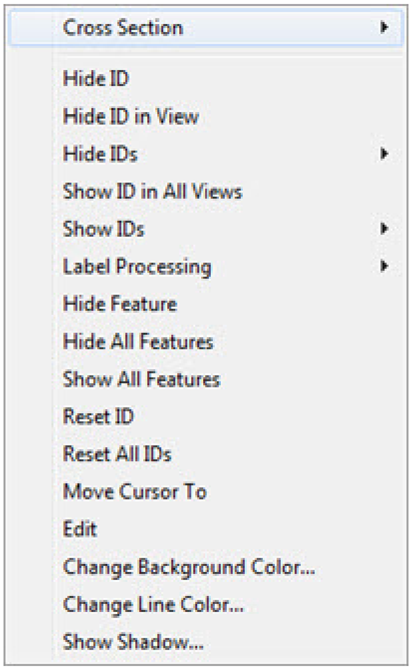
HINT: If a cross section is not selected, the first one below the current cursor position in the Edit Window is selected. Consequently, nothing happens if there are no cross sections defined below the current cursor position. The same occurs if the *last* cross section in the list is selected and this button is clicked.

Click the **Cross Section Slide Show** button a second time to exit the slide show and get the CAD graphic (3D view) back.

Show or Hide Cross Section Polylines from the Graphic Display Window

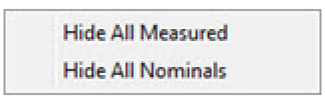
To hide cross section polylines from the Graphic Display window:

1. Right click a label of any cross section feature in the Graphic Display window to display a pop-up menu.

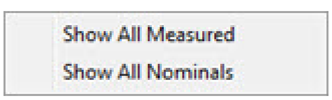


2. Hover over the **Cross Section** option to display the **Cross Section** menu.

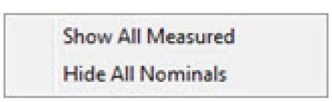
   If the measured and nominal cross section polylines are visible, the **Cross Section** menu has these options:

   If the measured and nominal cross section polylines are NOT visible, the **Cross Section** menu has these options:

   You may also have a mixture of the above options depending on the visible state of the polylines, such as:

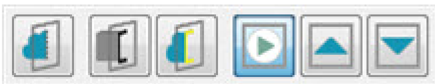
3. Click the appropriate option to show or hide the associated polylines.

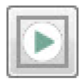
Measuring Cross Section Distances

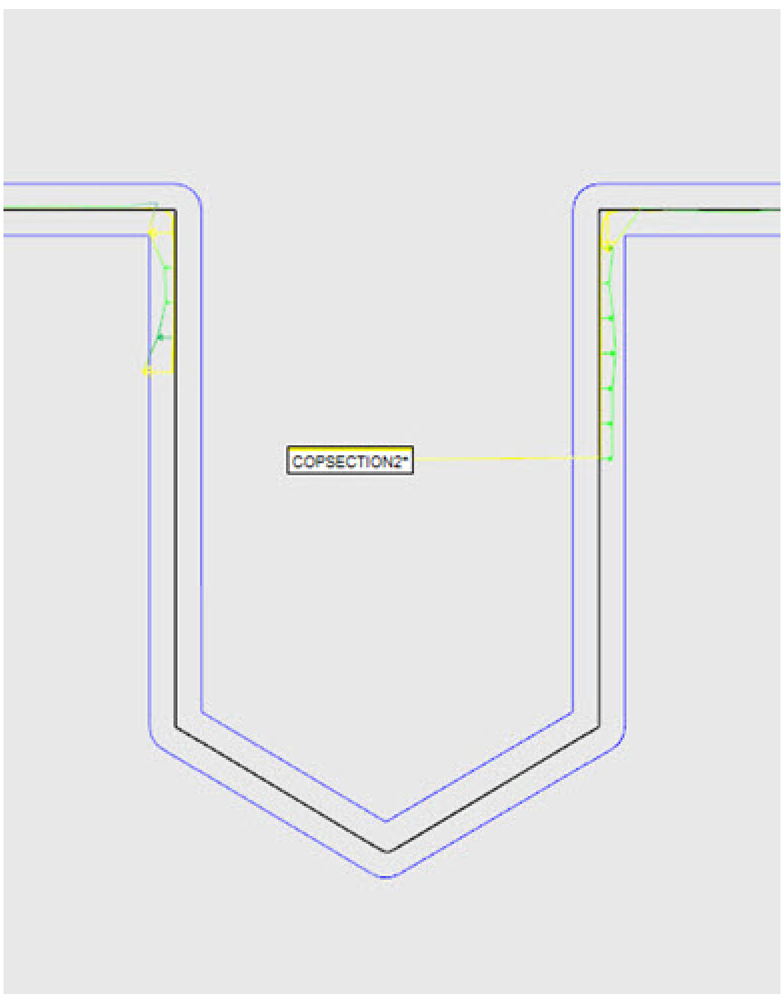
Distances can be measured on 2D cross sections in the Graphic Display window. You must already have the cross sections created and be in cross section 2D view. For details on how to view cross sections in 2D view, see "Show and Hide Cross Section Polylines".

To create a cross section Distance Gage:

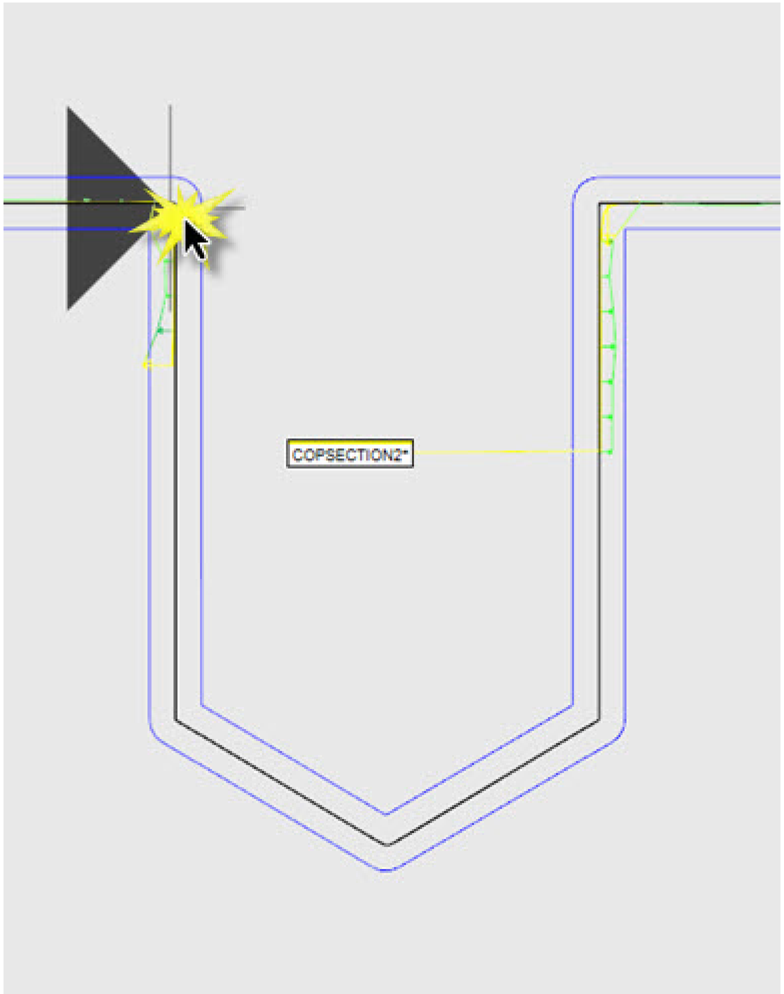
1. Once you have created the cross sections, from the Pointcloud or QuickCloud toolbar (**View | Toolbars**), click the **Cross Section** drop-down arrow to display the **Cross Section** toolbar.



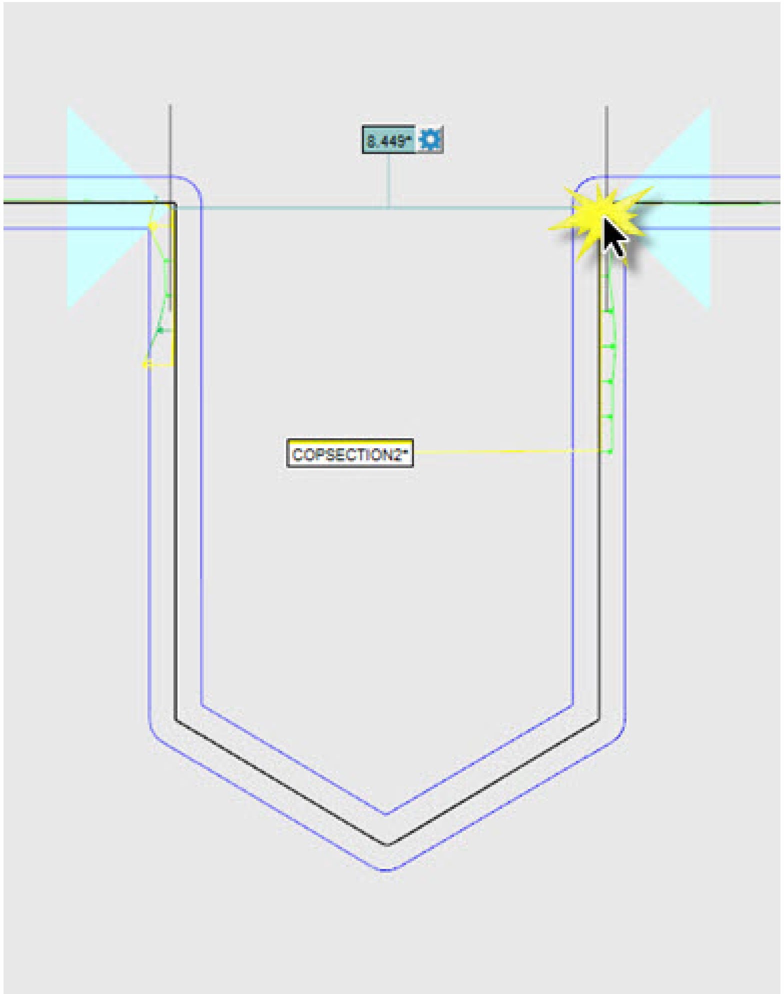
2. Click the **2D Slide Show** button  to enter 2D view.
3. Click the **Show the Previous Cross Section** or **Show the Next Cross Section** button until the cross section is displayed in the Graphic Display window.



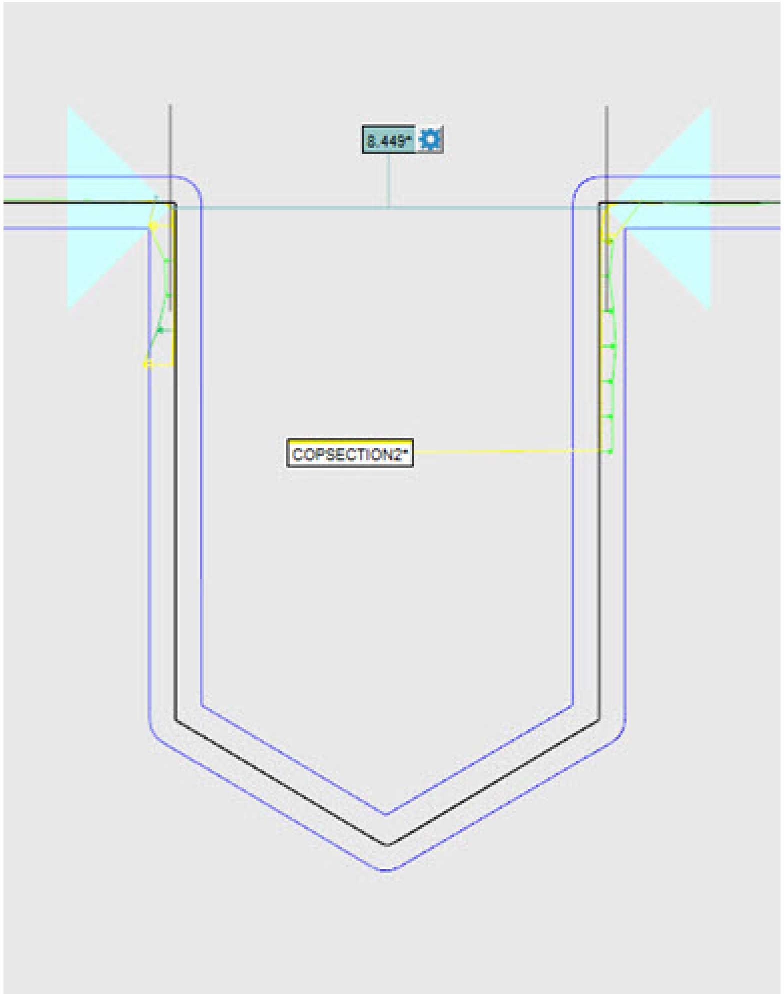
4. In the Graphic Display window, hover over the cross section, then click and drag to display the start point.



5. Drag the cursor to the end point and click to select it. The Distance Gage is calculated, created and displayed in the 2D view with its associated label.



As you drag the cursor, the software intuitively detects if the start and end points are along an axis. If so, the direction is recognized and is constrained parallel to that axis.



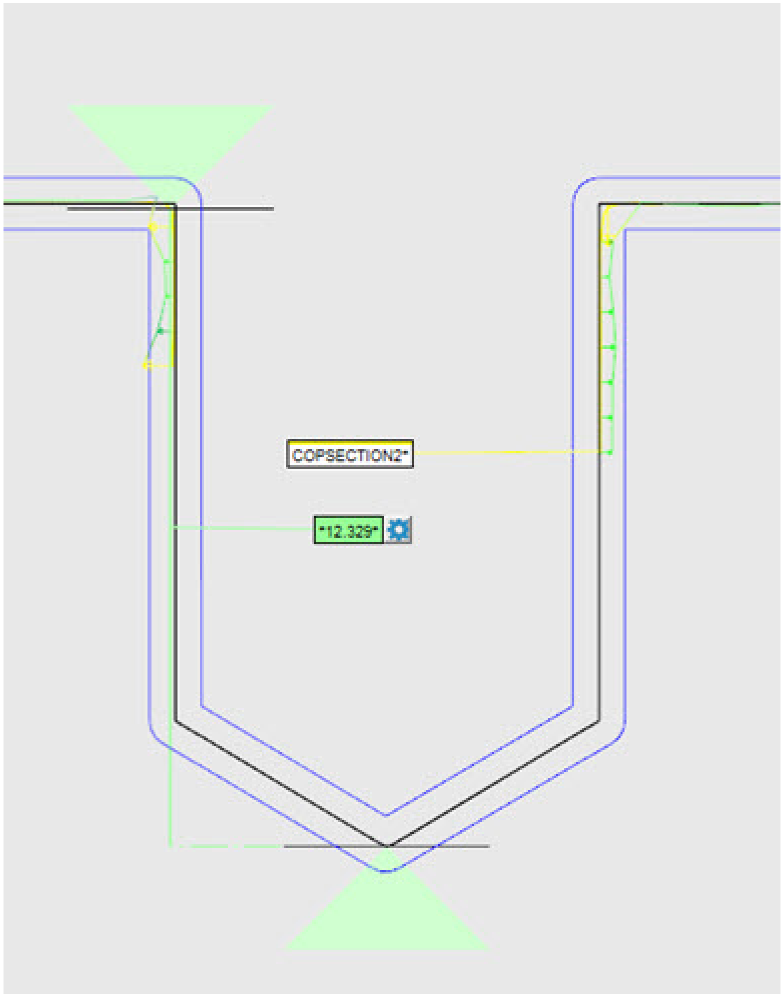
Parallel Distance Gage example

To create a Distance Gage parallel to the first side picked:

a. Press and hold the Shift key.
b. Click the start point and drag and click the end point.

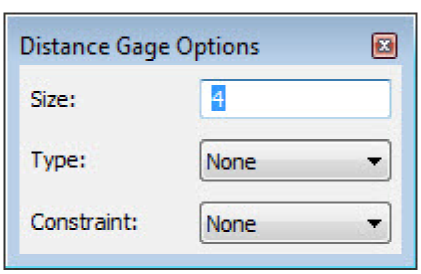
An example of this would be if the cross section was not created along the X, Y or Z axis.

If the start and end points are offset from one side to another, the axis direction is still recognized. The distance however, is calculated parallel but between the offset points.

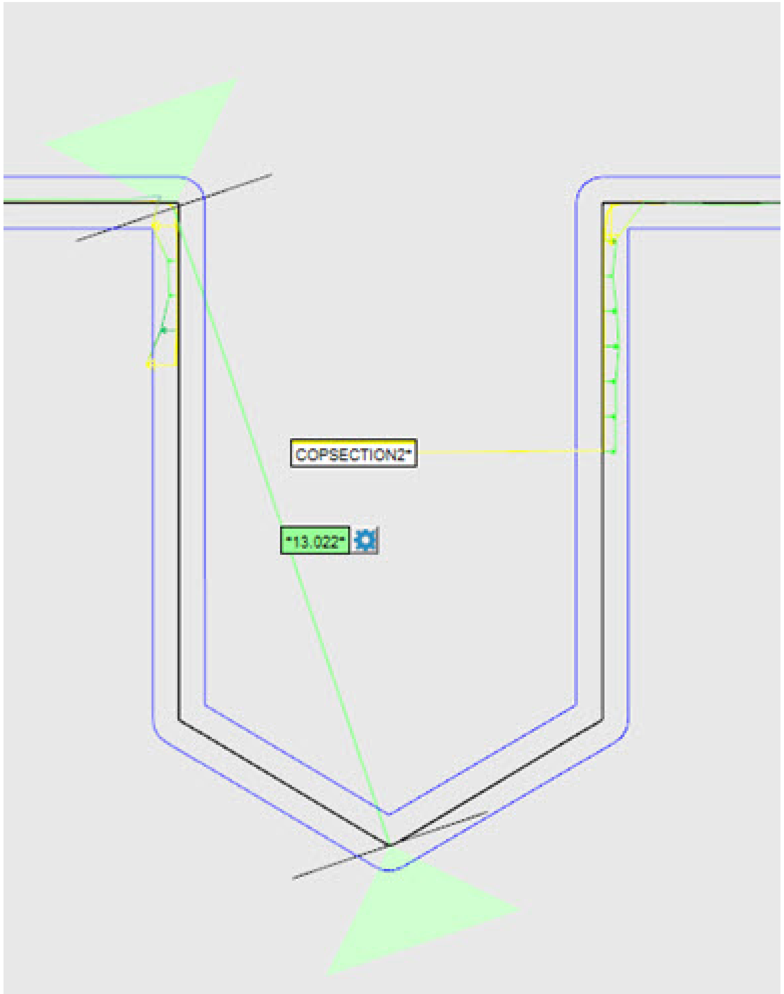


Offset Distance Gage example

6. To change properties of the Distance Gage, click the **Distance Gage Options** button on the label. The **Distance Gage Options** dialog box appears.



For example, if you do not want the Distance Gage calculated as an offset calculation, select the **Parallel** option from the **Constraint** list. Click the start and end points as before and the Distance Gage is calculated between the two points.



Example of a Distance Gage calculated with the Parallel Constraint option selected

7. Edit the properties of the Distance Gage:

   Size - If the **None** option is selected in the **Type** list, the **Size** value is used to determine the size of the start and end point icons in the Graphic Display window. If either the **Best Fit**, **Max Fit** or **Min Fit** options are selected from the **Type** list, the **Size** value is used as described below. A size of 4 is the default.

   Type - Click the drop-down arrow to display these options:

   - **None** (default) - A Point-to-Point distance calculation between the closest cross section polyline points based on the selected start and end points.
   - **Best Fit** - A Least Square line is calculated based on all the yellow points within the first pick zone, defined by the **Size** value (the default is 4) and the selected start point. This is repeated for the second pick zone, defined by the **Size** value and the selected end point. The centroid of the first Least

Square line is projected onto the measurement zone line. This is repeated for the centroid of the second Least Square line. The distance is between these two projected points.

- **Max Fit** - Defined by the point farthest out in the first pick zone, defined by the **Size** value and the selected start point, and by the point farthest out in the second pick zone, defined by the **Size** value and the selected end point. The Max Fit points are projected onto the measurement zone line. The Max distance is between these two projected points.
- **Min Fit** - Defined by the closest point in the first pick zone, defined by the **Size** value and the selected start point, and in the second pick zone, defined by the **Size** value and the selected end point. The Min Fit points are projected onto the measurement zone line. The Min distance is between these two projected points.

If the **Type** option is changed, the measured distance is automatically recalculated and the updated value is displayed based on the option selected.

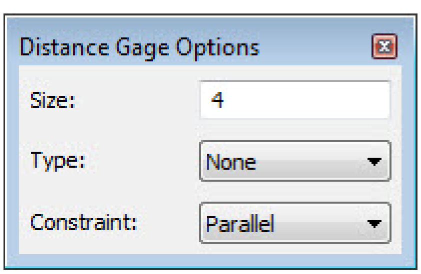
Constraint - Select **None** (default) if you do not want to constrain it to any axis. Select the appropriate option to constrain the Distance Gage to the **X**, **Y** or **Z** axis, or **Parallel** to calculate the distance parallel to the first selected side.

Creating a 3D Distance Gage

To create a 3D Distance Gage that is not constrained to any axis:

1. Press and hold the Ctrl key, then hover over the cross section in the Graphic Display window, and click and drag to display the start point.
2. Continue to drag the cursor with the Ctrl key pressed to the end point location.
3. Click to select the end point and display the Distance Gage and its associated label.

The same functionality is available as described earlier for 2D Distance Gages. Click the **Distance Gage Options** button to view the **Distance Gage Options** dialog box. The **Constraint** option is set to **None**.

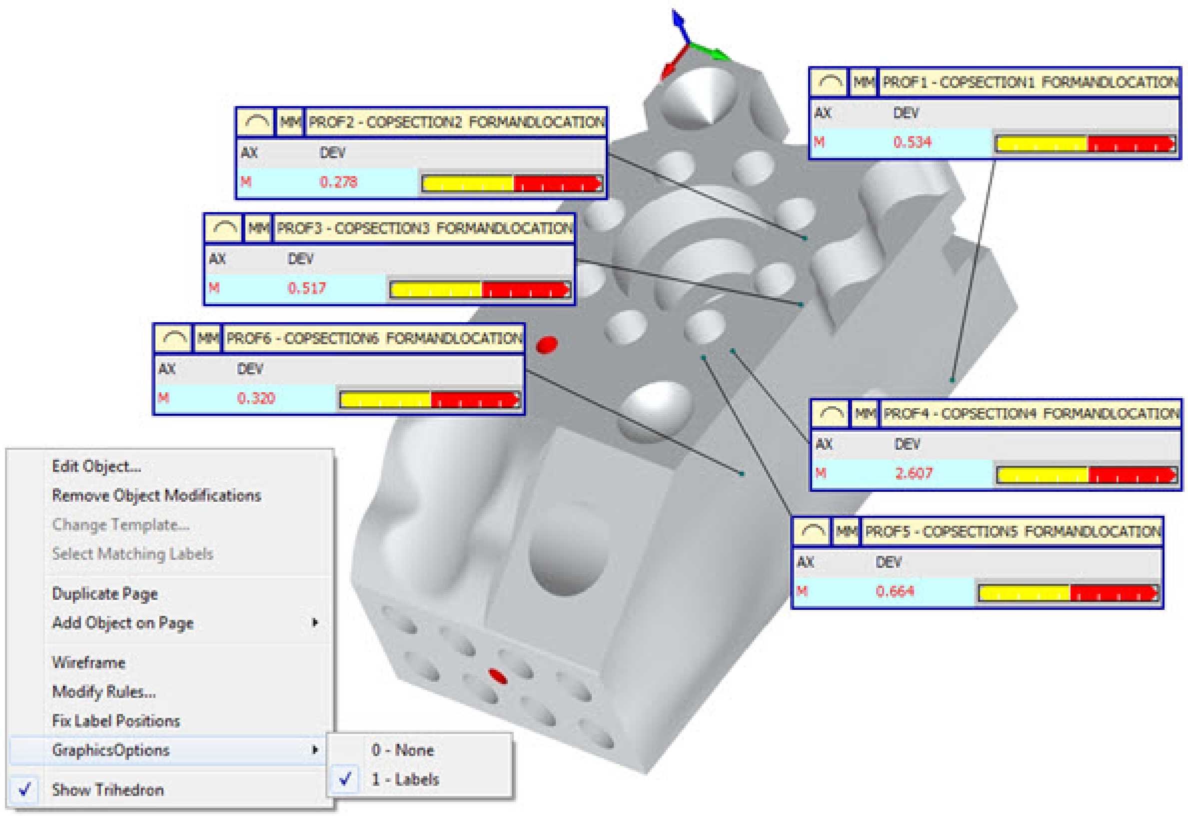


Viewing Cross Section Labels in Reports

You can view cross section Annotation and Distance Gage labels in reports in two ways:

Viewing labels from a report template that has a graphic image

1. From any report template that has a graphic image, right-click the image to display a pop-up menu.



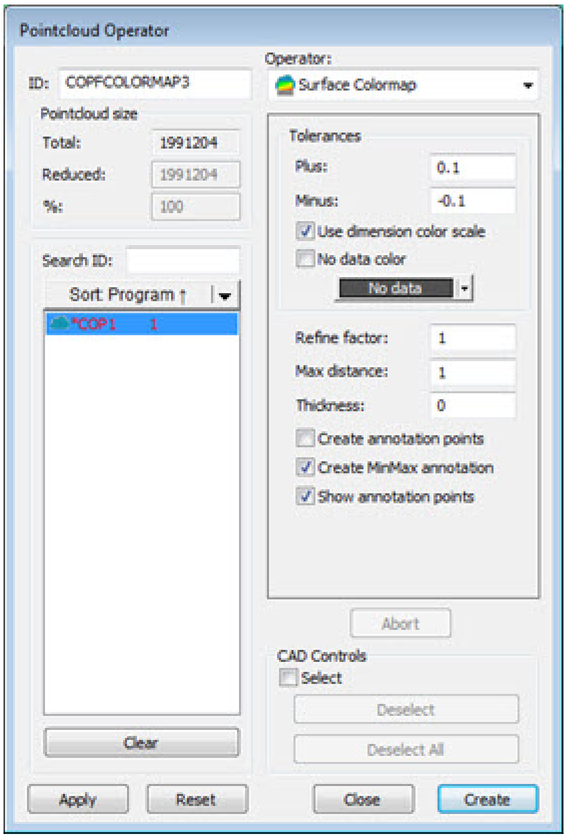
2. Click **GraphicOptions** then **1 - Labels** to display all the labels in your report. Click **0 - None** to hide all labels.

Viewing labels in the Report Graphical Analysis template from the Cross Section dialog box

1. Create the **Annotations** and **Distance Gage** items for your cross sections. For details on creating **Annotations**, see the "Cross Section" help topic. For details on creating **Distance Gage** items, see the "Measuring Cross Section Distance" help topic.

2. Create the Analysis View. For details on the `Analysis View` command, see the "Analysis View" description in the "Cross Section" help topic.
3. Click the **Graphical Analysis** option in the **Report** window (**View | Report**). The annotation and gage labels are automatically visible.

SURFACE COLORMAP

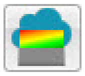


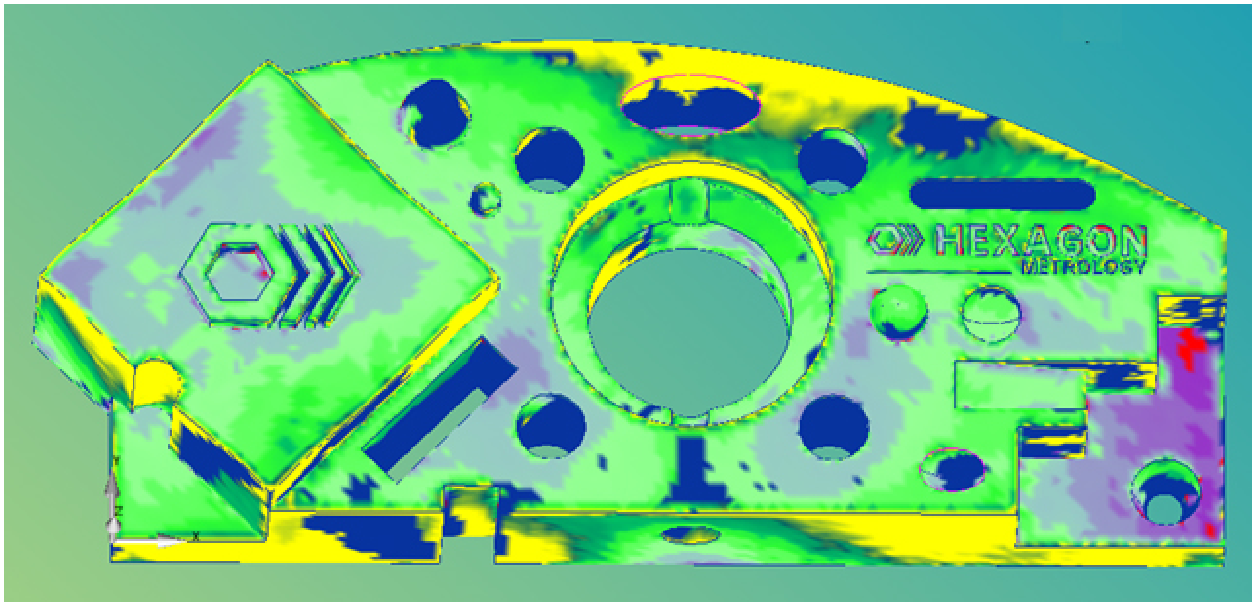
Pointcloud Operator dialog box - SURFACE COLORMAP Operator

The SURFACE COLORMAP operation applies a colored shading to the CAD model. The model is shaded according to the deviations of the cloud of points compared to CAD using the colors defined in the **Edit Dimension Color** dialog box and the tolerance limits specified in the **Upper tolerance** and **Lower tolerance** boxes discussed below.

The colors used for the colormap are defined in **Edit Dimension Color** dialog box (**Edit | Graphic Display Window | Dimension Color**).

You can view the color scale from the **Dimensions Colors** window by selecting **View | Other Windows | Dimensions Colors**.

 To apply the SURFACE COLORMAP operation to a Pointcloud, click **Pointcloud Surface Color Map** on the **Pointcloud** toolbar, or select **Insert | Pointcloud | Surface Colormap**.



Example of a Surface Colormap applied to selected CAD elements

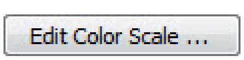
The SURFACE COLORMAP operator uses the following options:

Tolerances - Used to set the upper (Plus) and lower (Minus) tolerance values:

Plus - The upper tolerance value

Minus - The lower tolerance value

Use dimension color scale check box - When clicked, the color bar used for the Surface Colormap color properties is defined by the Dimension Color Scale color bar. For details on the Dimensions Color bar, see the "Using the Dimensions Colors Window (Dimensions Color Bar)" topic in the "Using Other Windows, Editors, and Tools" chapter of the PC-DMIS Core documentation.

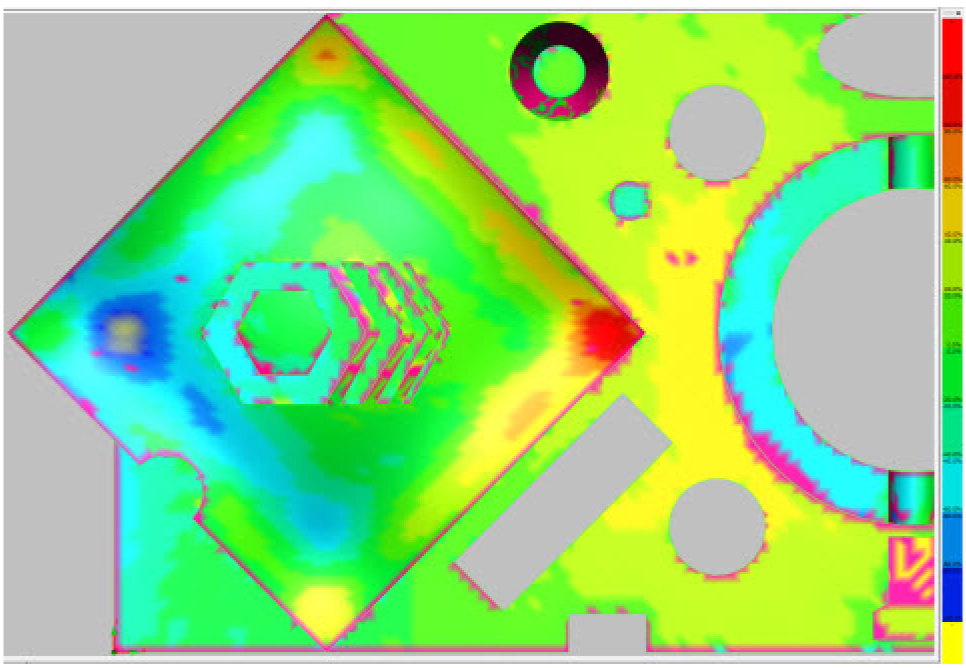


Edit Color Scale - When the **Use dimension color scale** check box is not marked, the **Edit Color Scale** button is enabled. When clicked, the functionality to dynamically change the color, scale and threshold of the surface and point colormap properties becomes available through the **Color Scale Editor** dialog box. See "Edit the Color Scale" topic for details.

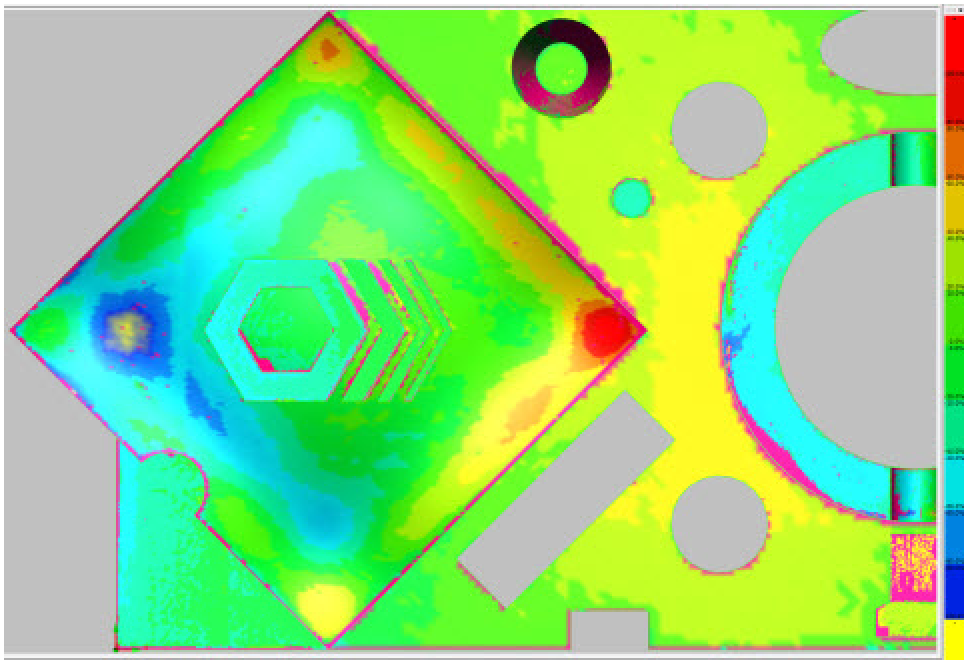
No data color check box - When you select this option, the specified color is mapped to selected surfaces where no data is defined.

Refine Factor - This adjusts the accuracy of the Surface colormap. If you change this value, PC-DMIS draws a new, changed colormap. The underlying measured data does not change. The colormap tessellates the CAD model with an overlay of colored triangles. The vertices of each triangle are colored with the color that corresponds to its deviation from the pointcloud. The colors are taken from the dimensions color scale discussed above. By using a smaller or larger refine factor value, you can generate a finer or coarser tessellation, respectively. You may want to decrease the refine factor to obtain a smoothly shaded CAD with a more accurate deviation representation. However, setting a smaller refine value results in a larger number of triangles, thereby increasing the computation time and the size of the CAD model. For comparison, note that the number of triangles for a refine factor of 0.5, compared to a refine factor of 1.0, is about 4 times more; whereas a refine factor of 0.1 compared to 1.0, is about 100 times more.

Sample Showing Refine Factor of 1:



Sample Showing Refine Factor of 0.1:



Max distance - This value only allows points that fall within the max distance to be included in the colormap. Note that if this value is too small, you may not see all the expected colored deviations. A good rule of thumb is to set this value slightly larger (10% for example) than the largest deviation.

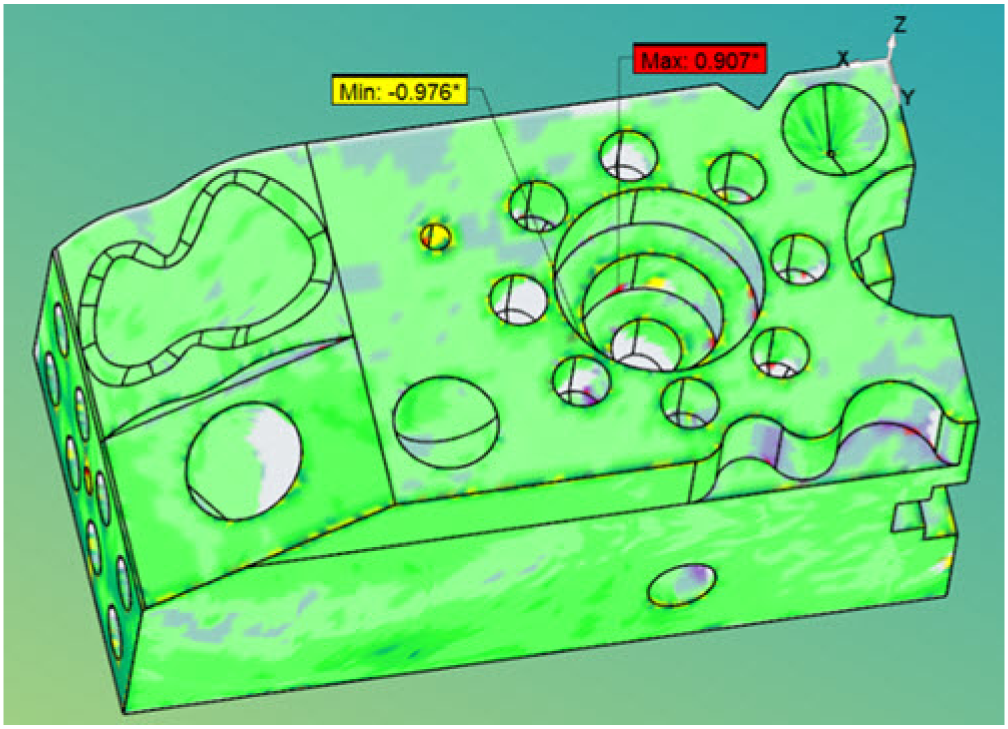
Thickness - This adds a thickness value to deviations on the colormap. This is useful if you want to add a material thickness to a CAD surface model.

Create annotation points check box - Annotations are a way to display the deviation for a specific location on a surface colormap with its associated color. To create an annotation:

1. Click the **Create annotation points check** box to mark it. This removes the check on the **Select** check box in the CAD Controls area and disable most of the options on the right side of the dialog.
2. Select a point on the CAD surface in the Graphic Display window. PC-DMIS evaluates and creates an annotation label in the same background color as the COP deviation point with the deviation value. The label can be moved around in the Graphic Display window as any other label.

NOTE: Once created, the annotation labels remain in the same position and have the same characteristics if the measurement routine is restarted, or if PC-DMIS is restarted and the same measurement routine is reloaded.

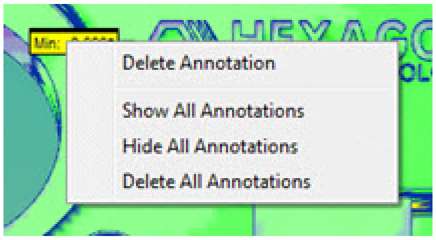
Create MinMax annotations check box - When marked, minimum and maximum values are created in the form of annotation labels for the active COP Surface Colormap.



The minimum and maximum points are re-calculated each time the measurement routine is executed.

Show, Hide, or Delete Annotation Labels

To show, hide, or delete annotation labels, right-click one to display the pop-up menu, then select the appropriate option.



Delete Annotation - The selected annotation label is automatically deleted.

Show All Annotations - All annotation labels are displayed.

Hide All Annotations - All annotation labels are hidden.

Delete All Annotations - All annotation labels are automatically deleted.

Show annotation points check box - When marked, any annotation points that have been created are displayed.

Click **Abort** to undo any calculations generated after clicking the **Apply** button.

CAD Controls - Lets you apply the operation to selected CAD elements. See "CAD Controls Area" where scanning is discussed for a more detailed description.

Clicking **Create** inserts a `COP/OPER,SURFACE COLORMAP` command into the Edit window.

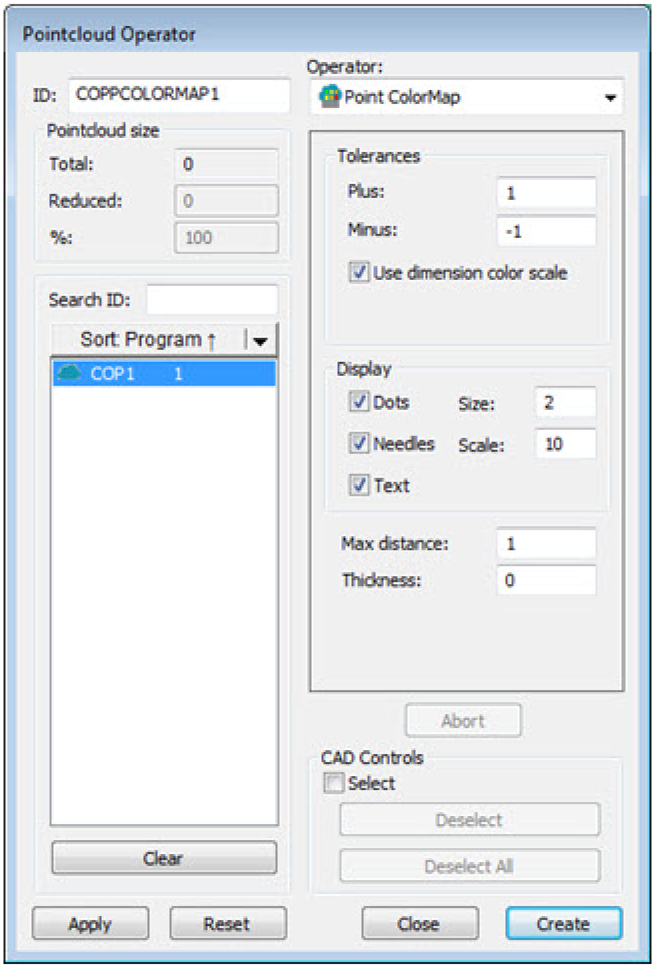
EXAMPLE: For example:

```
COPFCOLMAP2=COP/OPER,SURFACE COLORMAP,PLUS TOLERANCE=0.25,MINUS
TOLERANCE=-0.25,THICKNESS=0

REF,COP1,,
```

Colormaps in the Report

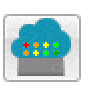
For information on how the software shows colormaps in the report, see the "Colormaps and the CadReportObject" topic in the "Reporting Measurement Results" chapter of the Core documentation.

POINT COLORMAP



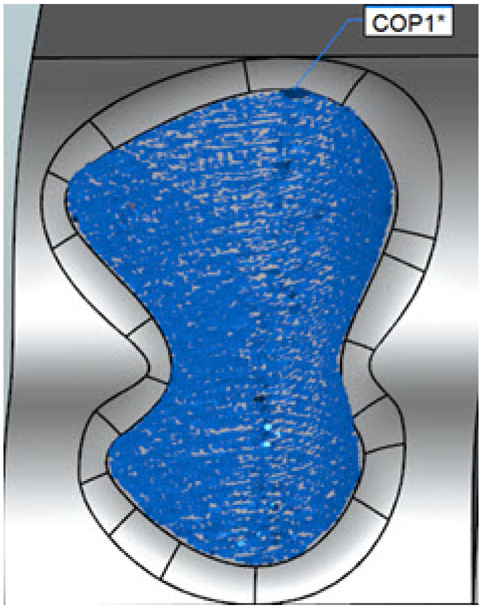
Pointcloud Operator dialog box - POINT COLORMAP Operator

The POINT COLORMAP operation evaluates the deviations of the data points contained in a COP command compared to a CAD object. The deviations can be represented by colored dots, colored needles showing the actual deviations or the numerical value of the deviations. The plus and minus tolerance, the size of the dots, the scale to be used for the needles, and the initial manual alignment need to be specified.

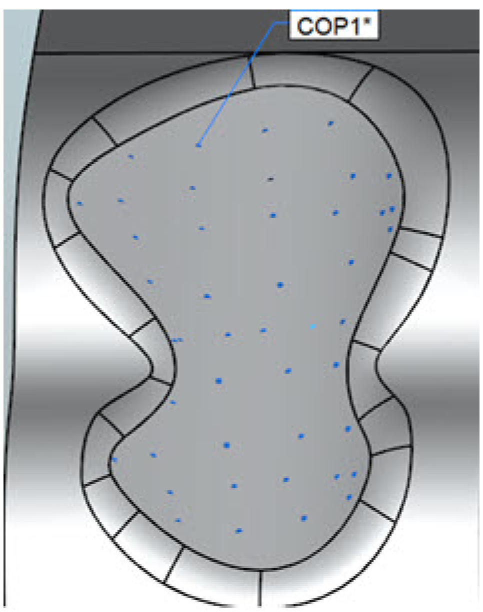
 To apply the POINT COLORMAP operation to a Pointcloud, click **Pointcloud Point Color Map** on the **Pointcloud** toolbar, or select **Insert | Pointcloud | Point Colormap**.

The recommended process when creating a point colormap is:

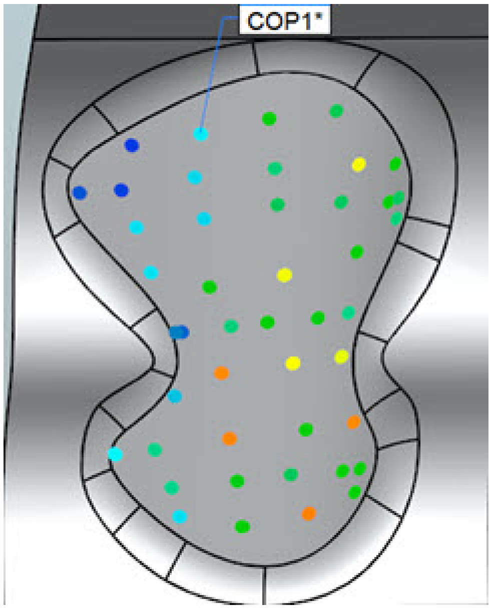
1. The data is cleaned or selected on the surface where the point colormap is needed.



2. Use the **DISTANCE** type setting from the **Filter** COP Operator to filter the data.



3. Create the point colormap.



Example of the recommended steps to apply a Point Colormap

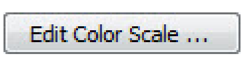
The Point Colormap operator has these properties:

Tolerances - Used to set the upper (Plus) and lower (Minus) tolerance values:

Plus - The upper tolerance value

Minus - The lower tolerance value

Use dimension color scale check box - When clicked, the color bar used for the Point Colormap color properties is defined by the Dimension Color Scale color bar. For details on the Dimensions Color bar, see the "Using the Dimensions Colors Window (Dimensions Color Bar)" topic in the "Using Other Windows, Editors, and Tools" chapter of the PC-DMIS Core documentation.



Edit Color Bar - When the **Use dimension color scale** check box is not marked, the **Edit Color Scale** button is enabled. When clicked, the functionality to dynamically change the color, scale and threshold of the surface and point colormap properties becomes available through the **Color Scale Editor** dialog box. See "Edit the Color Scale" topic for details.

Dots - Colored dots

Size - Size of the dots

Needles - The scaled deviation (using the **Scale** value below) as a colored line segment normal to CAD

Scale - Scale value to be used for the needle representation

Text - The numerical value of the deviation

Max distance - This value only allows points that fall within the max distance to be included in the colormap. Note that if this value is too small, you may not see all the expected colored deviations. A good rule of thumb is to set this value slightly larger (10% for example) than the largest deviation.

Thickness - This allows you to add a thickness value to deviations on the colormap. This is useful if you want to add a material thickness to a CAD surface model.

Click **Create** to insert a `COP/OPER,POINT COLORMAP` command into the Edit window.

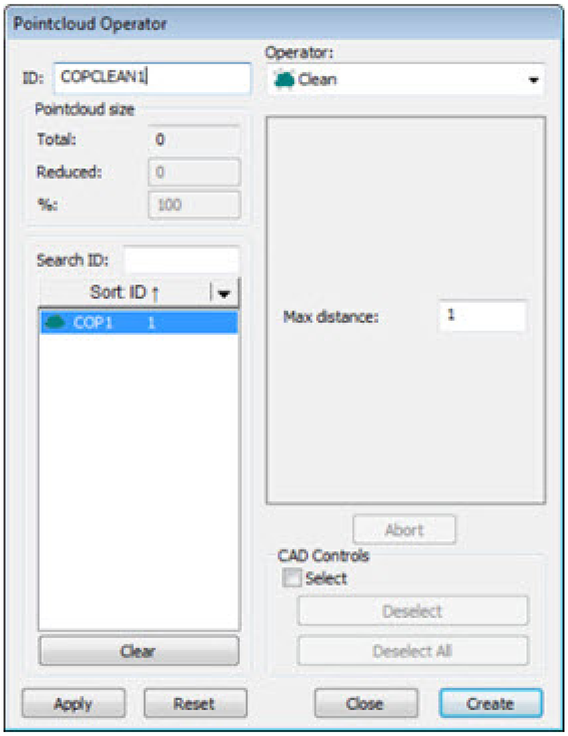
EXAMPLE: For example:

```
COPPCOLMAP1=COP/OPER,POINT COLORMAP,PLUS TOLERANCE=0.0394,MINUS
TOLERANCE=-0.0394,THICKNESS=0,

SHOW DOTS=YES,DOT SIZE=0.0787,SHOW NEEDLES=YES,NEEDLE
SCALE=10,SHOW LABELS=YES,

SIZE=50023

REF,COP2,,
```

Colormaps in the Report

For information on how the software shows colormaps in the report, see the "Colormaps and the CadReportObject" topic in the "Reporting Measurement Results" chapter of the Core documentation.

CLEAN



Pointcloud Operator dialog box - CLEAN Operator

The CLEAN operation is used to eliminate outliers by using the distance of the points to the CAD model of the part. If the distance of a point is greater than the value of MAX DISTANCE, the point is considered an outlier or not belonging to the part. To use this operation, you must have at least a rough alignment established (see "Creating a Pointcloud/CAD Alignment").

 To apply the CLEAN operation to a Pointcloud, click **Clean Pointcloud** on the **Pointcloud** toolbar, or select **Operation | Pointcloud | Clean**. This immediately cleans the point cloud.

If you select **Insert | Pointcloud | Operator** and then choose CLEAN in the **Operator** list in the **Pointcloud Operator** dialog box that appears, you can use the following options:

Max Distance - Indicates the maximum distance of a point to the CAD model for which the point is considered an outlier.

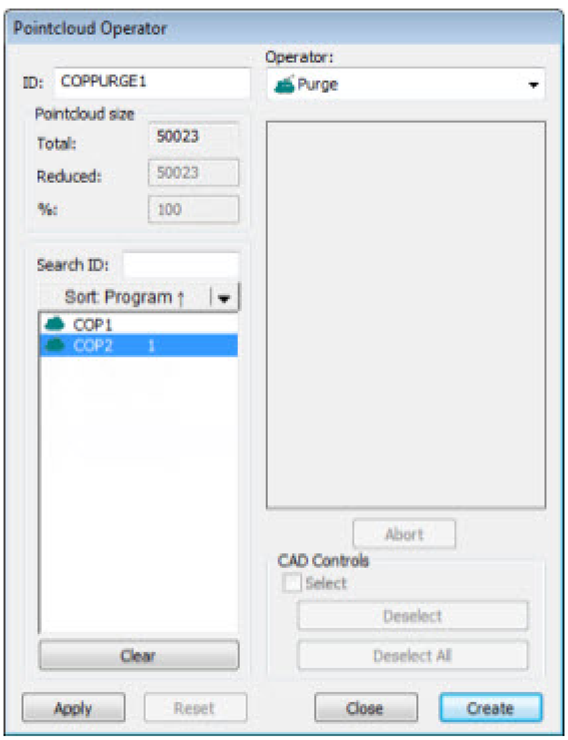
CAD Controls - Marking **Select** in this area lets you select the surfaces in the Graphic Display window around which the clean operation is based. The selected surfaces will be highlighted in red. The operation affects the entire cloud of points with respect to the selected surfaces. Any point that is located at a distance greater than the specified **Max distance** from all selected surfaces will be discarded. For example, suppose you select a single surface and type a value of 10. This means that any points in the COP located 10 or more units away from the selected surface will be cleaned. Any points in the COP within the length of 10 units of the selected surface will remain.

Clicking **Create** after editing the command inserts a `COP/OPER,CLEAN` command into the Edit window.

EXAMPLE: For example:

```
COPCLEAN4=COP/OPER,CLEAN,MAX DISTANCE=0.0399,SIZE=50023

REF,COP1,,
```

PURGE



Pointcloud Operator dialog box - PURGE Operator

From the COP command referenced by this operator, the PURGE operation removes all data points that do not belong to this operator. It is irreversible and affects all other operator commands that refer to same COP container so use with caution.

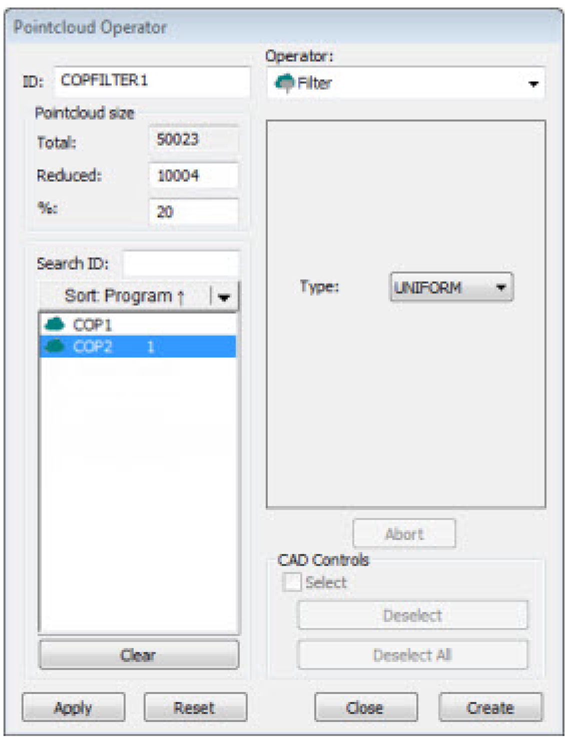
 To apply the PURGE operation to a Pointcloud, click **Pointcloud** on the **Pointcloud** toolbar, or select **Operation | Pointcloud | Purge**.

Clicking **Create** inserts a `COP/OPER,PURGE` command into the Edit window like the following examples:

```
COPPURGE1=COP/OPER,PURGE,SIZE=0
    REF,COPSECTION1,,
```

CAUTION: Once this command has been applied to a COP, there is no way to restore COP data that was removed. Undo does not restore this data.

FILTER



Pointcloud Operator dialog box - FILTER Operator

The FILTER operation filters data to a smaller subset of points.

 To apply the FILTER operation to a Pointcloud, click **Filter Pointcloud** on the **Pointcloud** toolbar, or select **Operation | Pointcloud | Filter**.

The Filter operator uses the following options:

Type - Indicates the type of filter operator to apply: **UNIFORM**, **CURVATURE**, **RANDOM**, **DISTANCE**, or **INCIDENCE ANGLE**.

UNIFORM - Generates a subset of points distributed evenly in the X, Y, and Z directions. It will produce the same effect as a regular grid in 2D, but in this case the effect is a 3D grid.

CURVATURE - Generates a subset of points with the highest estimated curvatures, mainly around edges, vertices, and highly curved areas of the surface.

RANDOM - Generates a subset of points randomly distributed in the pointcloud.

DISTANCE - Generates a subset of points where points are at least the specified **Distance** value apart from each other.

Distance - When **DISTANCE** is selected, the value entered specifies the distance for the distance filter.

INCIDENCE ANGLE - Generates a subset of points which excludes (i.e., filters out) points which have a normal vector orientation that fall outside the specified angle, relative to the laser sensor orientation. This filter allows you to remove laser points caused by secondary reflections or “noise”. You can see the effect of this filter after you click the **Apply** button from the dialog screen.

A valid value is any real number from 10 to 90 inclusive.

To use this filter, the pointcloud data must have vector information.

To filter COP data:

1. Select a filter type from the **Type** list.
2. From the list of commands, select the Pointcloud command to which you want to apply the filter.
3. Specify the number of points or percentage of points to keep after applying the filter in the **Reduced** or **%** boxes. This does not apply to the **Distance** filter.
4. Click the **Apply** button.

PC-DMIS filters the data, and the Graphic Display window shows the result. The size of the filtered data may differ slightly from the value that you specified. It is even more noticeable when the measurement routine is executed and the data is collected from scan commands. It is generally impossible to get the same number of points from a laser sensor that is repeatedly scanning the same entity.

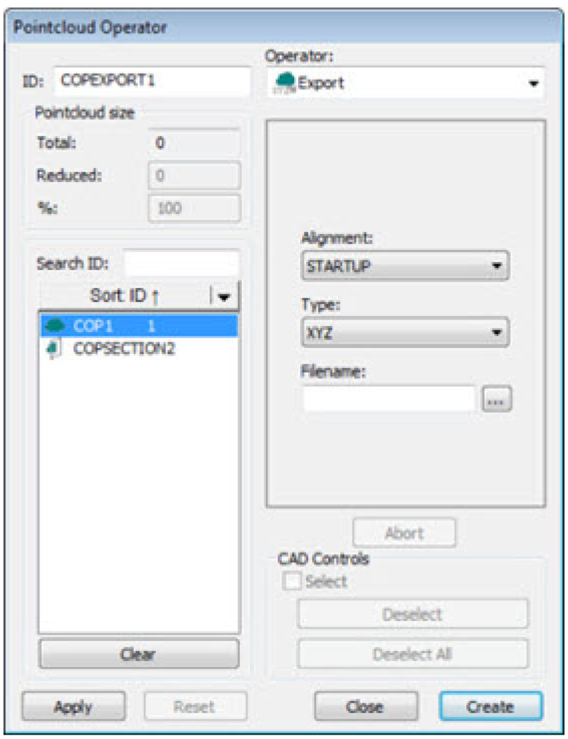
5. When you are satisfied with the result, click the **Create** button. PC-DMIS adds a `COPFILTER` command to the measurement routine containing all the information regarding the filter that you just applied.

Clicking **Create** inserts a `COP/OPER,FILTER` command into the Edit window like the following examples:

```
COPFILTER3=COP/OPER,FILTER,UNIFORM,SIZE=3000

    REF,COP1,,
```

In the example above, if the initial size of COP1 was 10,000 points, the filter replaces the 10,000 points held in COP1 with the filtered 3,000 points, so that COP1 now holds the filtered 3,000 points for its cloud of points. PC-DMIS flags the 7000 points it didn't use, so that you can undo the filter operation, using the RESET operation. Or if you desire, you can permanently purge the 7000 points not used using the PURGE operation. For more information, see "RESET" and "PURGE".

EXPORT



Pointcloud Operator dialog box - EXPORT Operator

The EXPORT operation exports the data in a COP or operator command in a specified format to an external file. The dialog of this operation is similar to the IMPORT operator.

To apply the EXPORT operation to a Pointcloud, click **XYZ**, **IGS**, or **PSL** on the **Pointcloud** toolbar, or select a menu option on the **File | Export | Pointcloud** menu.

The EXPORT operator uses the following options:

Alignment - Indicates the type of alignment to include when exporting the data.

Type - Indicates the type of format to which data is exported. It can be **XYZ**, **IGES**, or **PSL** (Polyworks).

Filename - Indicates the name of the export file.

Clicking **Create** inserts a `COP/OPER,EXPORT` command into the Edit window.

EXAMPLE: For example:

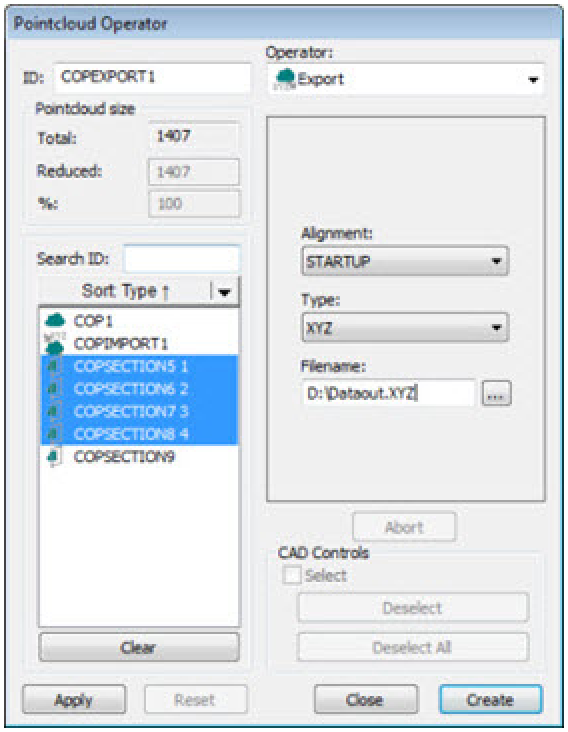
```
COPEXPORT1=COP/OPER,EXPORT,FORMAT=IGES,FILENAME=D:/Dataout.IGS,S
IZE=1623201

REF,COP1,,
```

Specify the format in FORMAT and the output file name in FILENAME, and then reference the COP command holding the data. If a filter has been applied to the COP command, then the `COPFILTER` command should be referenced for export rather than the original COP command.

EXAMPLE: For example, `REF, COPFILTER1,` rather than `REF, COP1,`. This ensures that the exported file reflects the filter set.

```
COPEXPORT2=COP/OPER,EXPORT,FORMAT=IGES,FILENAME=D:/Dataout.IGS,S
IZE=0

REF,COPFILTER1,,
```

It is also possible to select more than one command in the list of commands to export them in a single operation:



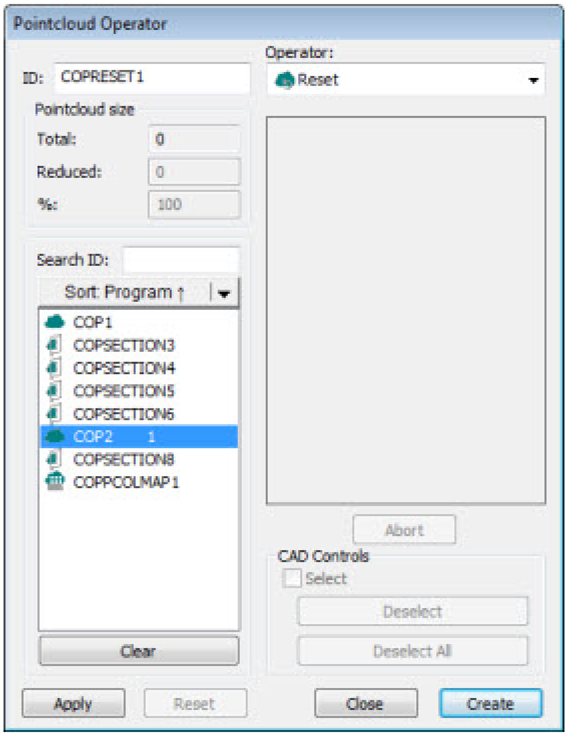
Pointcloud Operator dialog box with multiple commands selected

In this case, the command is inserted into the Edit window.

EXAMPLE: For example:

```
COPEXPORT1=COP/OPER,EXPORT,FORMAT=XYZ,FILENAME=D:/Dataout.XYZ,SI
ZE=1246

REF,COPSECTION2,COPSECTION3,COPSECTION4,COPSECTION5,,
```

RESET



Pointcloud Operator dialog box - Reset Operator

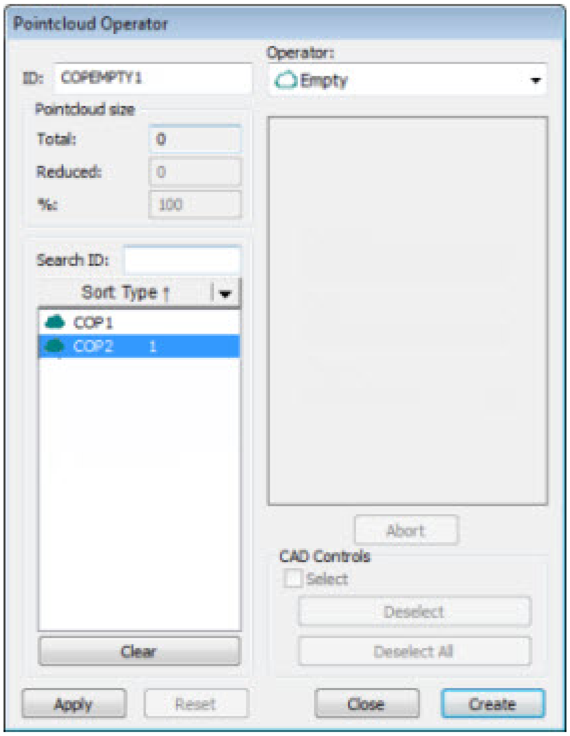
The RESET operation has a behavior similar to Undo and resets the data referred to in a previous operator command so that the new operator command represents all the data of the referred COP command and not only a subset.

 To apply the RESET operation, click **Reset Pointcloud** on the **Pointcloud** toolbar, or select **Operation | Pointcloud | Reset**.

Clicking **Create** inserts a `COP/OPER,RESET` command into the Edit window like the following examples:

```
COPRESET7=COP/OPER,RESET,SIZE=0
    REF,COPFILTER 2,,
```

EMPTY



Pointcloud Operator dialog box - EMPTY Operator

This operation deletes all of the data contained in a selected COP or operator command. When this command is executed, PC-DMIS removes the data of the associated COP.

 To apply the EMPTY operation to a Pointcloud, click **Empty Operation** on the **Pointcloud** toolbar, or select **Operation | Pointcloud | Empty**.

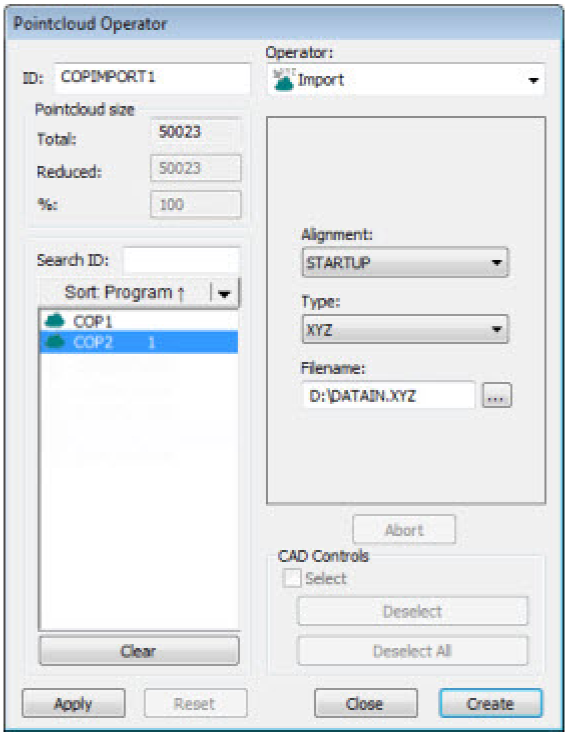
Clicking **Create** inserts a `COP/OPER,EMPTY` command into the Edit window.

EXAMPLE: For example:

```
COPEMPTY2 =COP/OPER,EMPTY,SIZE=0

REF,COP2,,
```

CAUTION: Once this command has been applied to a COP, there is no way to restore COP data that was removed. Undo does not restore this data.

IMPORT



Pointcloud Operator dialog box - IMPORT Operator

The IMPORT operation imports data from an external file into a COP command in the specified format. The dialog box for this operation is similar to the EXPORT operation.

To apply the IMPORT operation to a Pointcloud, click **XYZ**, **PSL**, or **STL** on the **Pointcloud** toolbar, or select a menu option on the **File | Import | Pointcloud** menu.

The Import operator uses the following options:

Alignment - Indicates the type of alignment to include when importing.

Type - Indicates the type of format from which data is imported. It can be **XYZ**, **PSL** (Polyworks), or **STL** type.

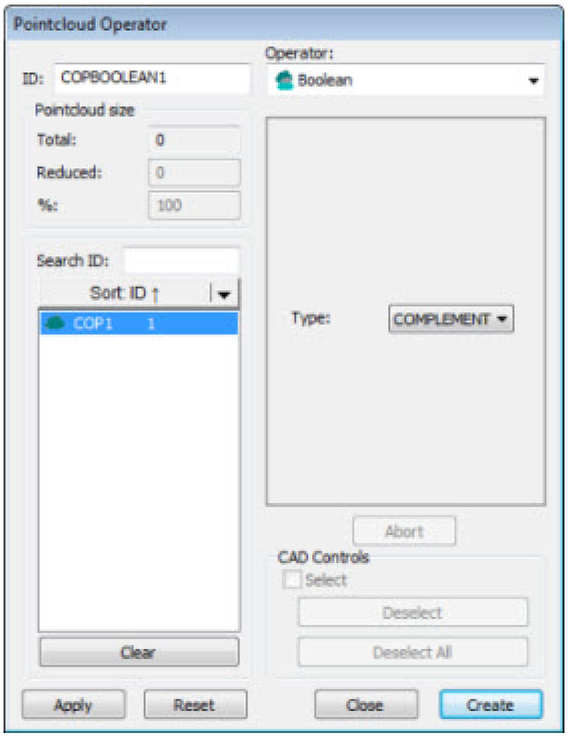
Filename - Indicates the name of the import file.

Clicking **Create** inserts a `COP/OPER, IMPORT` command into the Edit window.

EXAMPLE: For example:

```
COPIMPORT1=COP/OPER,IMPORT,FORMAT=XYZ,
FILENAME=D:/DATAIN.XYZ,SIZE=0

REF,COP1,
```

BOOLEAN



Pointcloud Operator dialog box - BOOLEAN Operator

This operation is applied on one or two selected operator or COP commands.

 To apply the BOOLEAN operation to a Pointcloud, click Pointcloud Boolean Operation on the Pointcloud toolbar.

The BOOLEAN operator uses the following option:

> **Type** - Indicates the type of Boolean operator to apply: **COMPLEMENT**, **UNITE**, **INTERSECT** or **DIFFERENCE**.
>
> > **COMPLEMENT** - This type generates the points that are not visible in a single selected command.
> >
> > **UNITE** - When applied to the two selected commands, this type generates a set of data points that contain all of the points in those commands.

INTERSECT - This type generates the set of data points that have the same locations in two selected commands.

DIFFERENCE - This type removes, from the first selected command, all of the points that are in common with the second selected command.

Clicking **Create** after editing the command inserts a `COP/OPER,BOOLEAN` command into the Edit window.

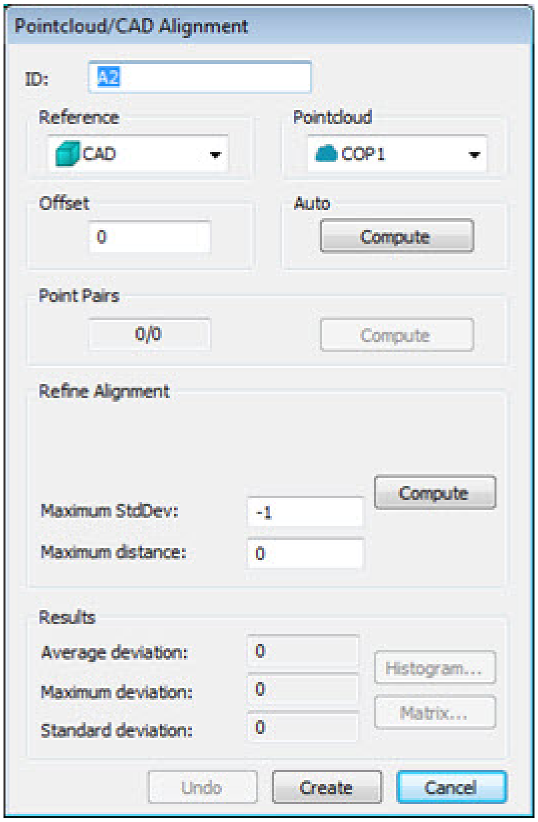
EXAMPLE: For example:

```
COPBOOELAN1=COP/OPER,BOOLEAN,UNITE,SIZE=0

REF,COPOPER2,COPOPER3,,
```

Pointcloud Alignments

In order to use the data you've collected in your clouds of points properly, you need to create an alignment between the pointclouds and the CAD data of your part model or between pointclouds. This is done using the **Alignment** dialog box.

Alignment Dialog Box Description



Default view of Pointcloud/CAD Alignment dialog box

ID - This displays the identification label for the alignment.

Reference - Select the point of reference for your alignment, usually either from the CAD itself or a defined COP.

Pointcloud - This list lets you choose the cloud of points to use in the alignment.

Offset - This defines an offset value for a surface CAD model and is typically used with sheet metal parts. Applying an offset value essentially gives the surface CAD model a thickness so you can align the pointcloud data to a different face that isn't represented in the surface CAD model. For example, if you have a surface CAD model for the top of a part but you want to align to a corresponding bottom surface, you could apply an offset value of the part's thickness to align the scanned data it to the bottom side. Use a positive value if you want to apply a thickness in the same direction as the surface normal vector; use a negative value if you want to apply a thickness opposite the surface normal. Only available for Pointcloud to CAD alignments.

Auto - This area lets you automatically align the CAD with the cloud of points by using the **Compute** button. Only available for Pointcloud to CAD alignments.

Point Pairs - This area lets you create a rough alignment based on selected points from the CAD that correspond to selected points from the pointcloud. Once you have the needed pairs selected, you can use the **Compute** button to perform the rough alignment.

Refine Alignment - This area allows for a more refined alignment. Only the **Maximum Distance** option is available for Pointcloud to Pointcloud alignments.

Depending on the alignment being made, the **Refine Alignment** area of the dialog box may consist of the following items:

> **IMPORTANT:** The first two options (**Total points** and **Maximum iterations**) are only available if PC-DMIS **IS NOT** setup to use the Reshaper SDK for alignment computations. For details on using the SDK for alignment computations, see the topic "`UseSDKForCopCadAlignments`" in the PC-DMIS Settings Editor documentation.

Total points - This box defines the number of random sampled points used to refine the alignment. This number must be a value of at least 3. A good number is around 200 points.

Maximum iterations - This box defines the number of repetitions the process will make in order to refine the alignment.

Compute - This button begins the refined alignment process. A progress bar on the status bar shows the progress as the process moves through the alignment iterations.

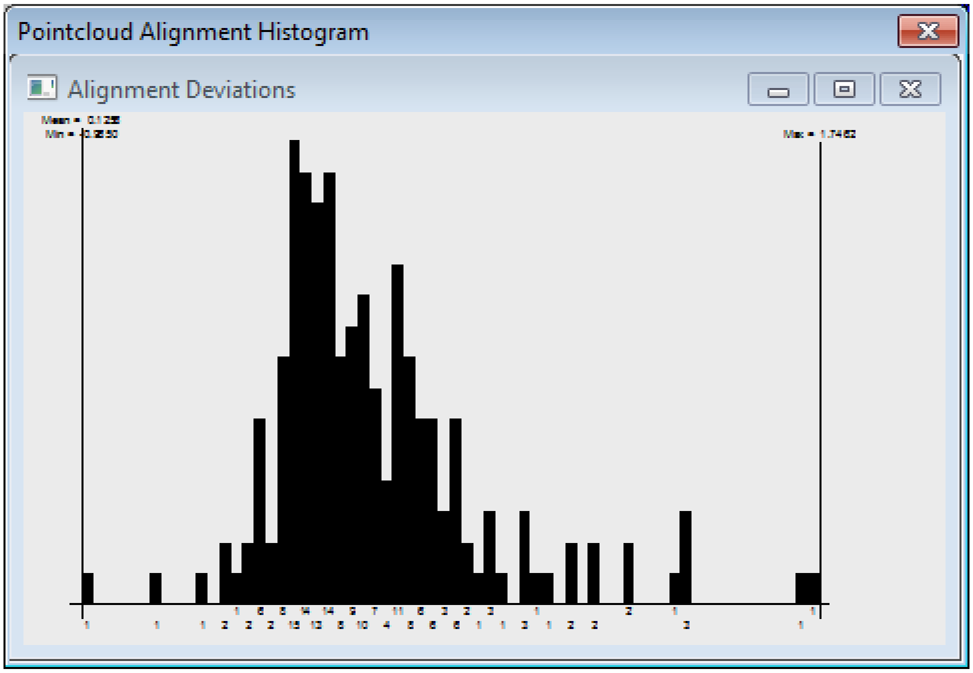
Maximum StdDev - Maximum StdDev is the maximum standard deviation used during the execution of an auto alignment. If the entered value is exceeded during the command execution, you are prompted to optionally pick point pairs on the CAD/Pointcloud. A value of -1 disables the Maximum StdDev functionality.

Maximum Distance - Defines the maximum distance PC-DMIS looks from the CAD for valid COP points. If no value is entered, the default value of 0 (zero) is used and the maximum distance becomes half the distance of the CAD bounding box.

Results - This area contains the following items:

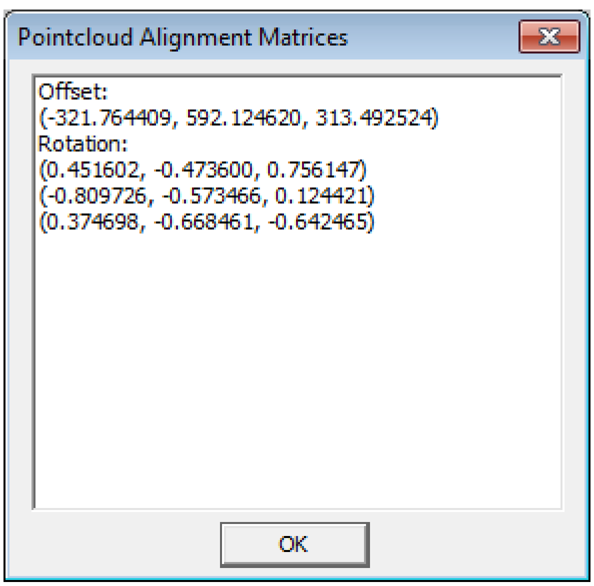
Information boxes showing the **Average Deviation**, **Maximum Deviation**, and **Standard Deviations** of the cloud of points in relation to the CAD model.

Histogram - This button takes a random sample of points from the point cloud, projects them onto the CAD, and then shows the deviations for that sample in the **Pointcloud Alignment Histogram** dialog box.



Sample Pointcloud Alignment Histogram dialog box

Matrix - This button displays the **Pointcloud Alignment Matrices** dialog box. This displays the numerical values of the alignment: the offset and the rotation matrix.

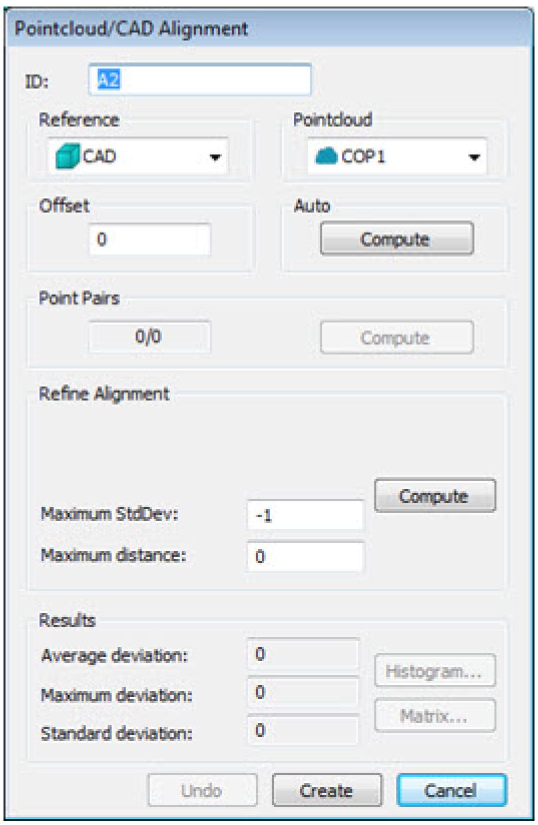


Pointcloud Alignment Matrices dialog box

Creating a Pointcloud/CAD Alignment

To create a Pointcloud to CAD alignment, do the following:

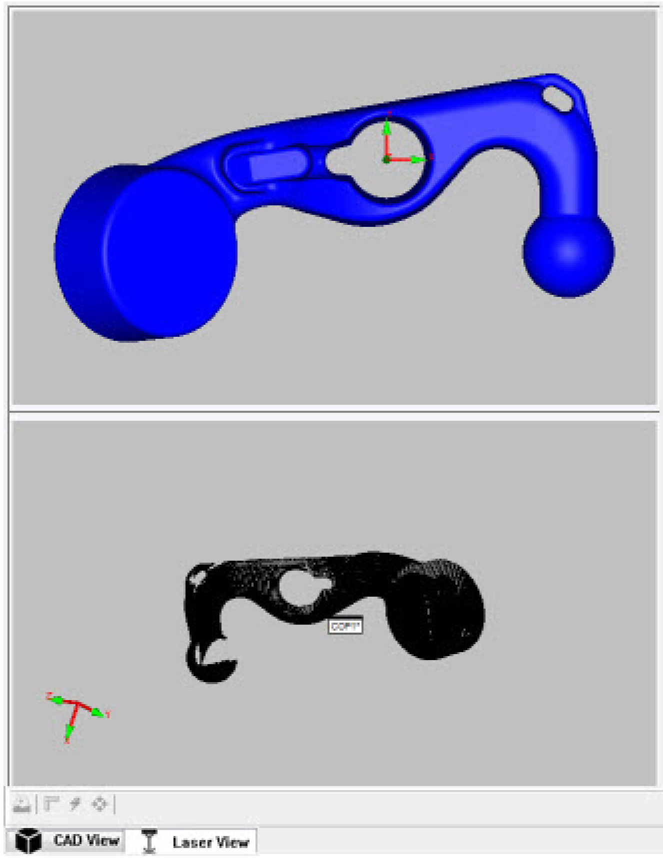
1. Ensure that you have an imported CAD model in the Graphic Display window and a COP command in the measurement routine. These elements are required to align pointclouds to the CAD.
2. Select the **Insert | Pointcloud | Alignment** menu option. You can also access this dialog box by typing the COPCADBF command in the Edit window's Command mode between the ALIGNMENT/START and the ALIGNMENT/END commands. The dialog box appears:



Pointcloud/CAD Alignment dialog box

> **NOTE:** For a complete description of the **Alignment** dialog box, see the "Alignment Dialog Box Description" topic in the PC-DMIS Laser documentation.

3. A temporary split screen view of the CAD model and the cloud of points appears in the Graphic Display window. You can use this split screen view to visually see the alignment taking place. Select your point of reference from the **Reference** drop down list - usually either the CAD model itself or a defined COP is available.

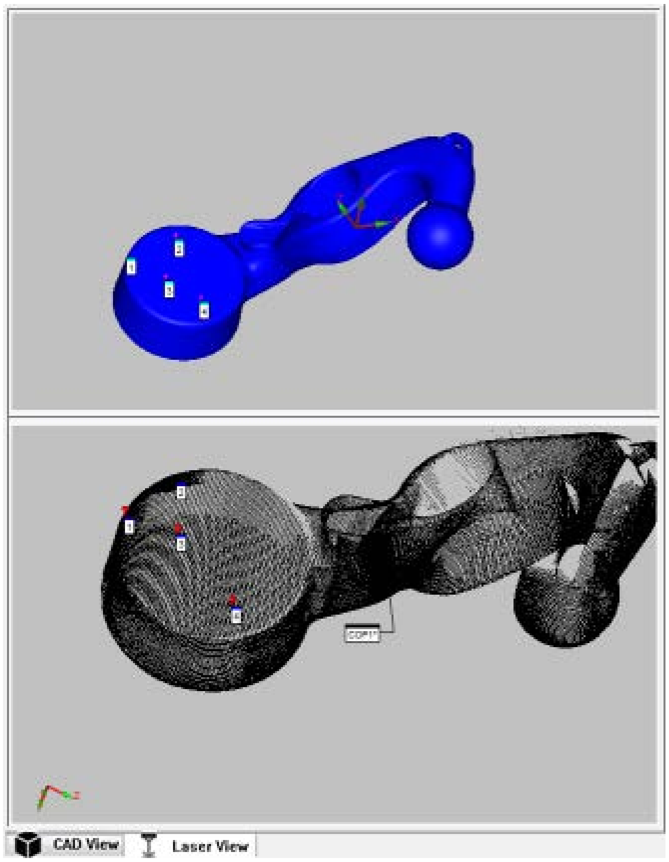


Split screen view showing the CAD model on the top view and the point cloud on the bottom view

4. If you have more than one pointcloud in your measurement routine, choose the cloud of points from the **Pointcloud** list.
5. Perform the alignment:

   - First, use the **Pointcloud/CAD Pairs** area to perform a rough alignment that brings the pointcloud close enough to CAD (if not already close) to be able to refine the alignment further if needed. You should use this type of alignment if the pointcloud is not complete or it contains scanned data belonging to a fixture, the table, and so forth.

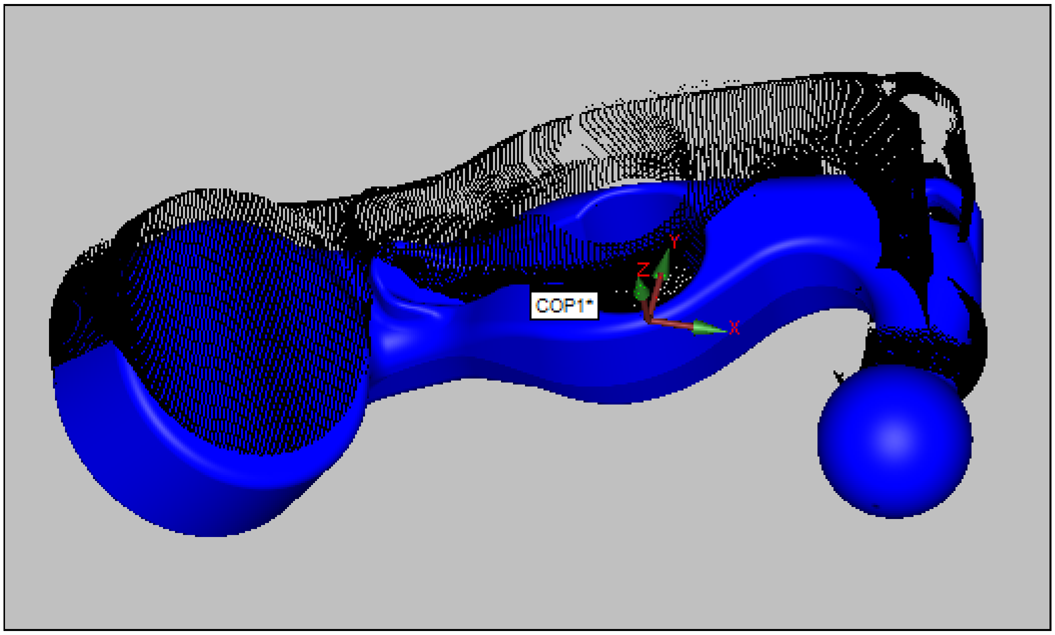
     - Click a desired number of points on the point cloud.
     - Click corresponding locations on the CAD model.



Split view showing selected cloud points and corresponding CAD points

- The more points you take around the different areas of the model and point cloud results in an improved alignment.
- Click **Compute** to create the rough alignment.

- Next, use the **Refine Alignment** area whenever you want to refine your alignment, thereby bringing the pointcloud closer to your CAD model. In order to be able to obtain a good refined alignment, the pointcloud points should be close enough to the CAD points through an initial rough alignment.



A sample rough alignment that needs a refinement

- Define the total number of random sample points to use in each iteration in the **Total Points**.
- Define the number of iterations in the **Maximum Iterations** box.
- Define the maximum standard deviation for the auto alignment execution between the points in the pointcloud and the CAD model using the **Maximum StdDev** box. When the auto alignment command is executed, if the standard deviation of Cop/CAD deviations is greater than the maximum value defined, you can select point pairs to get a better alignment. The default value is -1, equivalent to an infinite allowed standard deviation.
- Define the maximum distance of the points from CAD in order to be used in the best fit routines. The default value is 0. In this case, an internal max distance based on the size of the pointcloud is used.
- Click **Compute** to refine the alignment.

- Alternately, you can use the **Auto Area** to automatically create the alignment. You should only use this when you have a clean pointcloud (without outliers) and a full scan of the external faces of the part. Simply click **Compute**. This also performs a refinement on the alignment as it generates.

6. If a portion of the cloud doesn't align nicely with the CAD, you can click the **Undo** button and recompute using the same type of alignment with additional parameters, or you can try a different alignment.

7. If you have a surface model representing a sheet metal part, and you want to align to the offset faces, define an **Offset** value representing the constant thickness of the sheet metal part.
8. Use the **Results** area to determine how well the point cloud aligned with the CAD. Make any changes to the **Offset** or **Refine Alignment** values to improve the alignment if necessary. If any changes are made, be sure to click the **Compute** button to regenerate the alignment with the new values.
9. Once you're satisfied with the alignment, click **Create**. PC-DMIS closes the temporary split screen view and places the `COPCADBF` command in the Edit window. See the "COPCADBF Command Mode Text" topic.

> **HINT:** If needed, you can adjust the `CadGridSizeForPointcloudCadAutoAlignment` registry entry to define the distance between the grid of points used to align the pointcloud to the CAD model.

COPCADBF Command Mode Text

The COPCADBF command allows you to perform a best fit alignment of clouds of points with your CAD data.

Below is an example code snippet for a COPCADBF alignment:

```
A1 =ALIGNMENT/START,RECALL:STARTUP,LIST=YES
    COPCADBF/REFINE=n1,n2,n3,n4,n5,SHOWALLPARAMS=TOG1,
    ROUGH ALIGNPAIR/
        THEO/<x,y,z>,<i,j,k>,
        MEAS/<x1,y1,z1>
    REF,TOG2,,
ALIGNMENT/END
```

n1 represents the total number of sample points to use in the refinement.

n2 represents the maximum number of iterations.

n3 represents the offset value for applying a thickness.

n4 represents the maximum standard deviation value.

n5 represents the maximum distance value.

TOG1 lets you show or hide the parameters used for the rough alignment. It can be set to YES or NO.

```
ROUGH ALIGNPAIR/
    THEO/x,y,z,i,j,k,
    MEAS/x1,y1,z1
```

These rough alignment pairs of points are defined/selected using the Graphic Display window. The values next to `THEO/` represent the point on CAD. The values next to `MEAS/` represent the corresponding point on the COP. These pairs are used to determine a rough transformation between the CAD and the COP which allows the COP to come close enough to the CAD to allow further refinements of the alignment.

TOG2 lets you choose the pointcloud to use for the alignment.

Creating a Pointcloud/Pointcloud Alignment

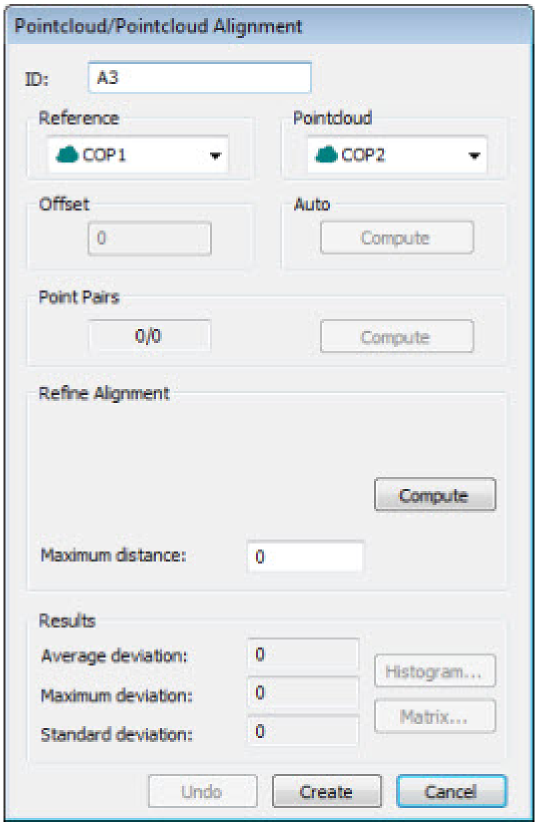
The Pointcloud to Pointcloud alignment functionality allows you to best fit align two pointclouds which have been collected in two different reference frames that have some overlap. A typical example is two scans in two pointcloud commands, representing areas of a part that cannot be scanned in the same part orientation.

The alignment is done in two steps:

- A rough alignment, where pairs of points in the overlapping area of the two clouds are selected
- A refined bestfit, which tries to bring the second cloud as close as possible to the reference cloud.

To create a Pointcloud to Pointcloud alignment, do the following:

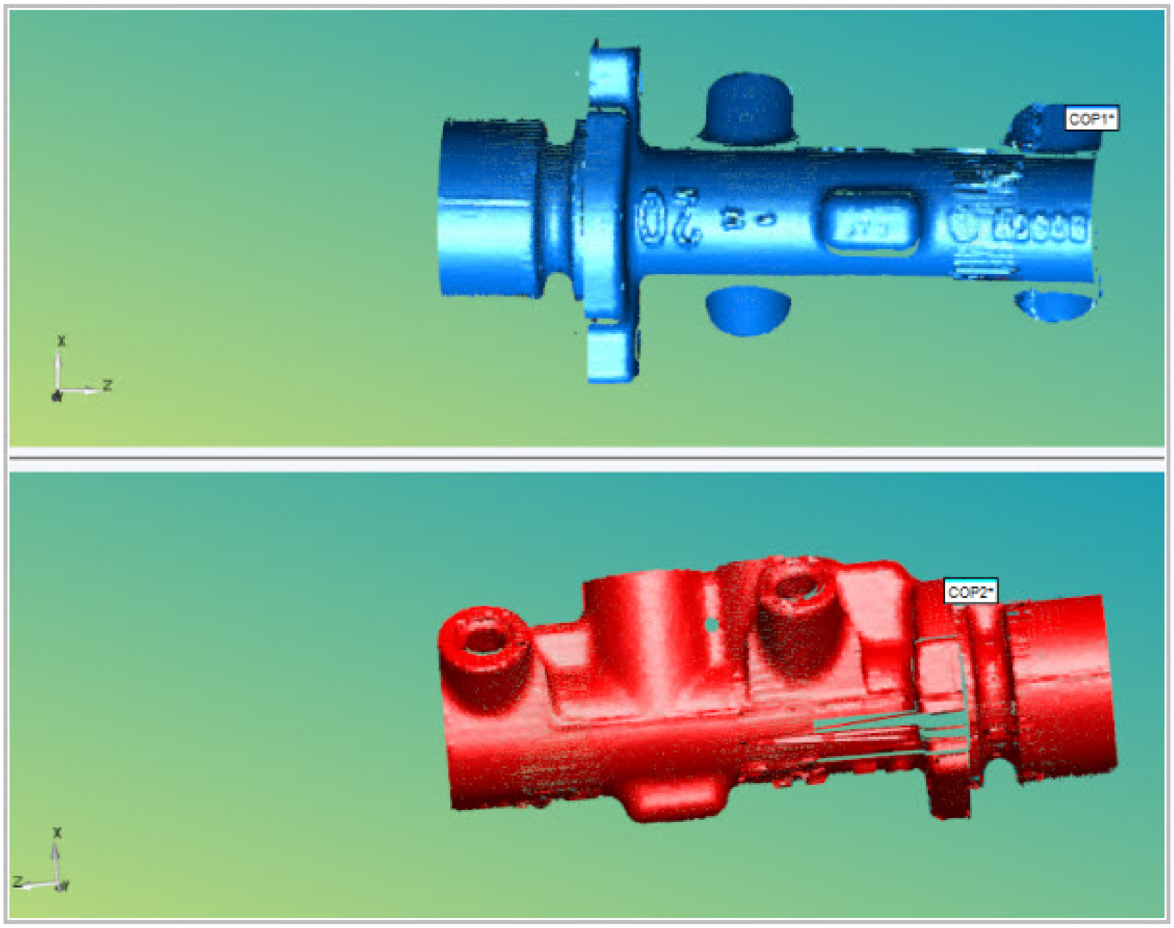
1. Ensure that you have two or more COP commands in the measurement routine that you are using to align. These elements are required to align two pointclouds.
2. Select the **Insert | Pointcloud | Alignment** menu option. You can also access this dialog box by typing the COPCOPBF command in the Edit window's Command mode between the ALIGNMENT/START and the ALIGNMENT/END commands. The dialog box appears:



Pointcloud/Pointcloud Alignment dialog box

HINT: For a complete description of the **Alignment** dialog box, see the "Alignment Dialog Box Description" topic in the PC-DMIS Laser documentation.

3. A temporary split screen view of the two cloud of points appears in the Graphic Display window. You can use this view to visually see the alignment taking place. Select your first COP to be used as a point of reference from the **Reference** drop down list.

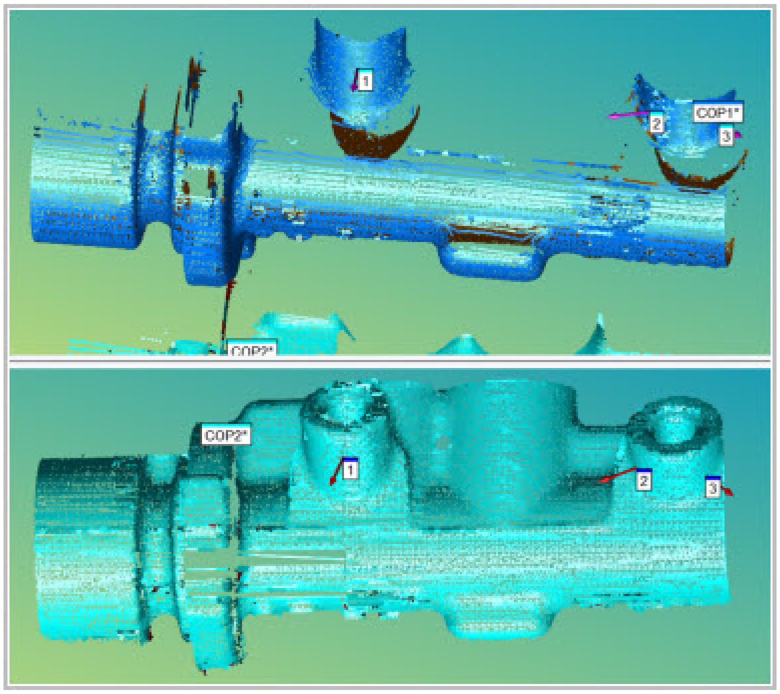


Split screen view showing a pointcloud to pointcloud alignment

4. Perform the alignment:

   - First, use the Point Pairs area to perform a rough alignment that brings the pointclouds close enough to each other. This is a mandatory step.

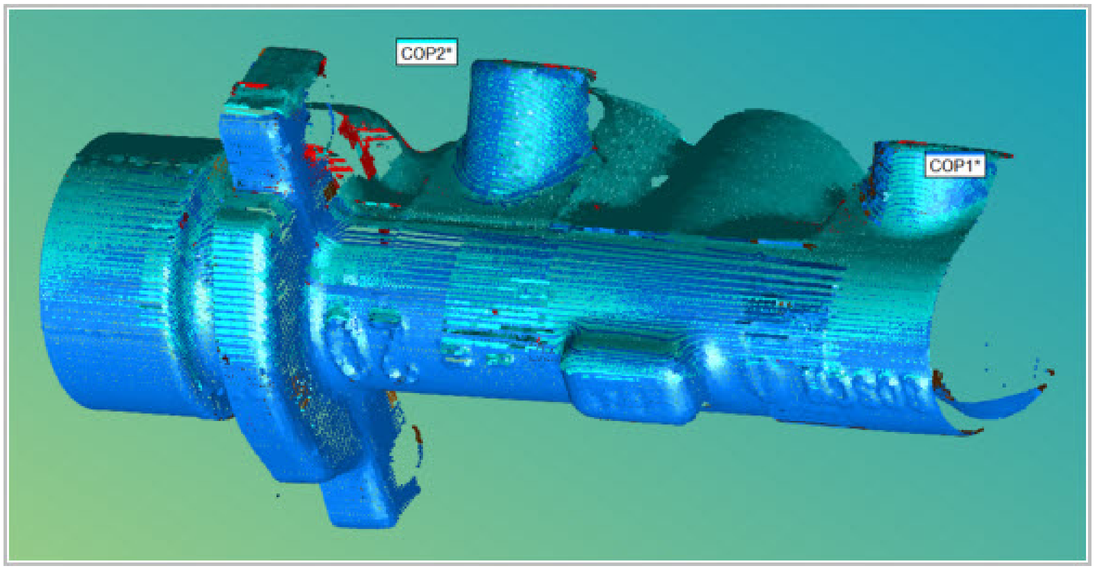
     - Click a desired number of points on each of the pointclouds. At least three pairs should be clicked ONLY on the overlap area of the two clouds.



Split view showing selected COP1 and COP2 pointclouds

- The more points you take around the overlap areas of the pointclouds results in an improved alignment. Click **Compute** to create the rough alignment.

- Next, use the Refine Alignment area whenever you want to refine your alignment, thereby bringing the two pointclouds closer to each other. In order to be able to obtain a good refined alignment, the two pointcloud points should be close enough to each other through the initial rough alignment.



A sample rough pointcloud to pointcloud alignment that needs a refinement

- Define the maximum distance between the points in the two pointclouds using the **Maximum Distance** box. The default value is 0 (zero). If the default value is used, PC-DMIS uses an internal default value related to the dimensions of the pointclouds.
- Click **Compute** to refine the alignment.

5. If a portion of one cloud doesn't align nicely with the other, you can click the **Undo** button and recompute using the same type of alignment with additional parameters, or you can try a different alignment.
6. Once you're satisfied with the alignment, click **Create**. PC-DMIS closes the temporary split screen view and places the `COPCOPBF` command in the Edit window. For details on the COPCOPBF command, see the "COPCOPBF Command Mode Text" topic in the PC-DMIS Laser documentation.

COPCOPBF Command Mode Text

The COPCOPBF command allows you to perform a best fit alignment of the reference clouds of points with a second cloud of points.

Below is an example code snippet for a COPCOPBF alignment:

```
A1 =ALIGNMENT/START,RECALL:STARTUP,LIST=YES
    COPCOPBF/REFINE,SHOWALLPARAMS=TOG1,
    ROUGH ALIGNPAIR/
```

```
        THEO/<x,y,z>,<i,j,k>,
        MEAS/<x1,y1,z1>
    REF,TOG2,TOG3,,
ALIGNMENT/END
```

TOG1 lets you show or hide the parameters used for the rough alignment. It can be set to YES or NO.

```
ROUGH ALIGNPAIR/
    THEO/x,y,z,i,j,k,
    MEAS/x1,y1,z1
```

These rough alignment pairs of points are defined/selected using the Graphic Display window. The values next to `THEO/` represent the point for the reference COP. The values next to `MEAS/` represent the corresponding point on the second COP. These pairs are used to determine a rough transformation between the reference COP and the second COP which allows the two cloud of points to come close to allow further refinements of the alignment.

TOG2 determines the reference COP used for aligning to the second COP.

TOG3 determines the second COP used for the alignment back to the reference COP.

TCP/IP Pointcloud Server

PC-DMIS can send your pointcloud data to a custom-built third-party software. It uses a TCP/IP communication protocol to do this. To establish the connection, your custom application must be able to load a dynamic link library (dll) file named PcDmisPointCloudClientDll.dll. You can request this file from Hexagon Customer Support.

Once your application loads the dll file, click on one of these TCP/IP Pointcloud server icons available from PC-DMIS's **Pointcloud** toolbar to establish the connection:

TCP/IP Pointcloud Server Connection with Local Copy - This establishes the connection with the client, sends the point cloud data directly to the client, and when the scan finishes, the point cloud data remains inside the measurement routine.

TCP/IP Pointcloud Server Connection without Local Copy - This establishes the connection with the client, sends the point cloud data directly to the client, and when the scan finishes, the point cloud data is deleted from the measurement routine.

Extracting Auto Features from Pointclouds

Laser Auto Features can be extracted from scanned Pointcloud data. Once the Auto Features are set up, this allows you to simply scan the part and Auto Feature information will be extracted from the scan. Multiple Auto Features can be included and extracted from a single Pointcloud.

Review the following topics to execute Auto Feature extraction from manual scans:

- Defining a Laser Auto Feature by Clicking on a Pointcloud
- Executing Scan-Extracted Auto Features
- Aligning Measured Auto Features to CAD

See "Laser Probe Toolbox: Feature Extraction tab".

Defining a Laser Auto Feature by Clicking on a Pointcloud

Often, users will define Auto Features by clicking on the CAD. In the case where no CAD exists, you can perform a scan of the part, and then click on the individual point cloud points to define your Auto Feature; or you can box select the feature from the point cloud.

To define an Auto Feature from Pointcloud points:

1. Scan the surface of the part in which the needed Auto Feature exists.
2. Click the needed Auto Feature from the **Auto Feature** toolbar or the **Insert | Feature | Auto** sub-menu. This opens the **Auto Feature** dialog box.
3. Either select points from the cloud of points that best define the feature's nominal position or drag a box directly on the cloud of points to have PC-DMIS extract the feature from the points within the dragged box. PC-DMIS will define the Auto Feature based on your selection.

Defining Features by Selecting Points

The following table shows the number of points that are needed to define an Auto Feature's location.

Feature	Points to Select
Surface Point	Select one point at the needed location within the measured surface area.
Edge Point	Select one point at the needed location along the measured edge.
Plane	Select at least three points that best define the needed plane's nominal position.
Circle	Select at least three points around the perimeter of the measured circle.
Round Slot	Select three points along one of the slot's arcs then select another three points along the other arc.
Square Slot	Type the slot's nominal **Width** in the **Auto Feature** dialog box. Select two points along a long side of the slot. Select one point on a short side of the slot. Select one point on the other long side of the slot. Finally, select one point on the other short side of the slot.
Flush and Gap	Select a point on each side of the gap.
Cylinder	Select three points for each of two circles that define the extents of the cylinder's form and length.
Sphere	Select at least five points around the surface of the measured sphere.

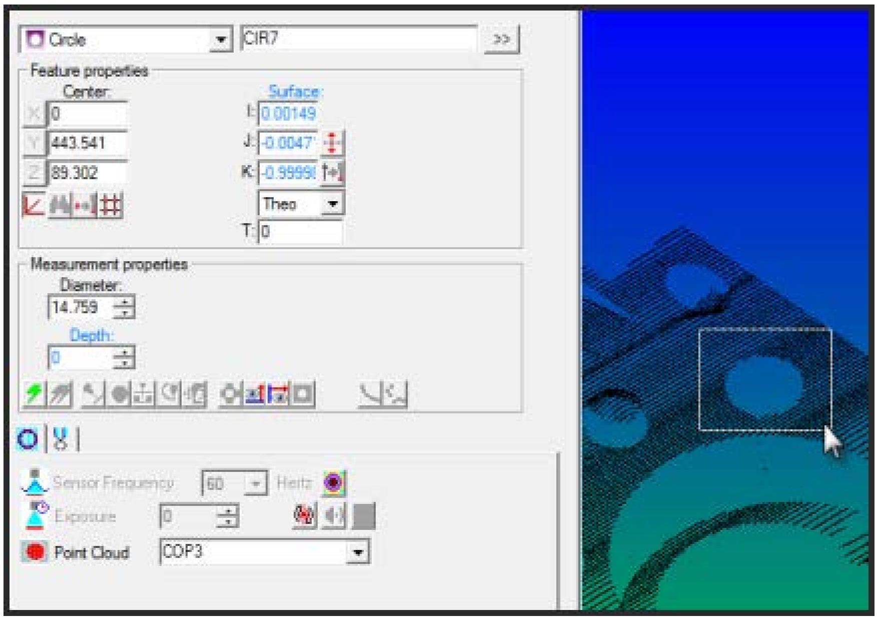
Defining Features by Box Selecting

During learn mode you can drag a box around the desired feature on the pointcloud to extract supported auto features using the selected data points.

This functionality has these limitations:

- PC-DMIS only calculates the surface vector. You may need to define the angle vector manually, such as for a polygon feature.

- If your box selection includes points at multiple depths in the Z axis, it may result in a poor feature extraction. You can avoid this by either clipping the acquisition or by using `COP/OPER,SELECT` to exclude those points prior to the box selection.



Example Circle Feature Creation by Box Selecting

This works with these supported features:

- Surface Point
- Plane
- Circle
- Round Slot
- Square Slot
- Sphere
- Polygon

For all other auto features, you must use the point selection method.

Executing Scan-Extracted Auto Features

When executing manual scans from which Auto Features are extracted, you should do the following:

1. Scan the Auto Features in you measurement routine in any order. You can do this with one or more passes. After the first pass, if the scan's Pointcloud points have changed for a feature, then the feature's measured values are recalculated.
2. When all the Auto Features associated with the scan have solved successfully, the command in the Edit Window is highlighted in yellow.
3. When Auto Features have solved and have been reported correctly, the command in the Edit Window is highlighted in green.
4. If additional scan data is taken for a feature that has already been solved, the feature's measured values are updated again with the new solution.
5. Once all of the included Auto Features have been solved, you may choose to continue scanning to further refine the measured results, or you can click the **Scan Done** button  from the **Execution** dialog box. You can also finish the scan by pressing the done button on your measurement arm.

HINT: The **Scan Done** button is not available until all the included Auto Features are successfully measured.

See the "Using Pointclouds".

Aligning Measured Auto Features to CAD

The procedure is only available when you measure Auto Features with a manual laser sensor (on a portable arm) and with imported CAD data. This allows you to select the *actual* measured features from the pointcloud that correspond to selected *nominal* features from the CAD.

To align measured auto features to CAD nominals:

1. Import CAD data.
2. Open the **Auto Feature** dialog box for a feature you want to include in the manual alignment.
3. Select the nominal location for the feature. Click on the CAD surface next to the feature to do this.
4. Change any auto feature parameters as needed and click **Create** to add the auto feature to the measurement routine.
5. Repeat steps 2 through 4 for each auto feature you want to include in the alignment.

NOTE: PC-DMIS automatically adds a new extraction COP when you begin to create a new laser auto feature. You may include the features of the manual alignment in the same pointcloud. The Laser Probe Toolbox: Laser Scan Properties tab determines the COP from which the software extracts the laser auto features.

6. Execute the measurement routine. PC-DMIS prompts you to scan the laser auto features as part of a Portable Laser Alignment.
7. Scan the part to include the auto features for the manual alignment. You may need to perform more than one scan to adequately define each feature.
8. When you have completed scans, on your measurement arm, press the **Done** button.
9. PC-DMIS now prompts you to define the first manual alignment feature. Follow the instructions provided in the dialog box and status bar and then click **OK**. At the end of selection the software displays the preliminary form of the auto feature.
10. Repeat Step 9 for each of the manual alignment features.

NOTE: PC-DMIS solves the laser auto feature with the theoretical values from the CAD and actual values from the measured pointcloud.

11. Select the **Insert | Alignment | New** menu item (Ctrl+Alt+A) to open the **Alignment Utilities** dialog box.
12. From the list box, select the alignment features, and click **Auto Align**. PC-DMIS aligns the defined features from pointcloud with the corresponding CAD nominals. This establishes the manual laser alignment.

Creating Auto Features with a Laser Sensor

With PC-DMIS Laser, you can use your laser sensor to create these auto features:

- Laser Surface Point
- Laser Edge Point
- Laser Plane
- Laser Circle
- Laser Slot
- Laser Flush and Gap
- Laser Polygon

- Laser Cylinder
- Laser Cone
- Laser Sphere

NOTE: This topic only discusses auto features as they pertain to laser sensor operations. For detailed information on auto features, consult the "Creating Auto Features" section in the main PC-DMIS documentation.

Implementation of Quick Features in PC-DMIS Laser

In order to cleanly implement the Quick Feature functionality, rules must be applied when switching between certain feature types that have the Inner/Outer options (Laser Circle, Laser Round Slot, Laser Square Slot, Laser Cylinder, Laser Cone and Laser Sphere for example).

NOTE: This is not available for Flush and Gap features since the hovering mouse functionality is not available for this feature type.

Since the Inner option enables LEAST_SQR and MAX_INSC and the Outer option enables LEAST_SQR and MIN_CIRCSC, the following rules apply:

- Whenever the Inner/Outer option selected in the dialog as default matches the Inner/Outer information that comes from the CAD quick selection, the best fitting algorithm default is kept in the created feature.
- When the Inner/Outer option selected in the dialog as default doesn’t match the Inner/Outer information that comes from the CAD quick selection, the best fitting algorithm default is kept in the created feature only if LEAST_SQR was set as a default. In all the other cases, the created feature will have the Inner/Outer information coming from the CAD and the best fitting algorithm option set to LEAST_SQR.

For example, if you set as default Outer Circle and best fitting algorithm as MIN_CIRCSC and then you quick select an inner circle, you’ll get an Inner Circle with LEAST_SQR option as result.

For more information on ways to create Quick Features, see the "Quick Ways to Create Auto Features" chapter of the PC-DMIS Core documentation.

Common Laser Auto Feature Dialog Box Options

In PC-DMIS Laser, the **Auto Feature** dialog box works alongside the **Probe Toolbox** to create a complete laser auto feature command. To edit an auto feature, you can use the Edit window and modify the command there, or you can change parameters inside the **Auto Feature** dialog box and the **Probe Toolbox**. See "Using the Laser Probe Toolbox:" for information on the toolbox.

The following **Auto Feature** dialog box options are common to all the supported Laser Auto Feature types and are discussed briefly for each of the dialog box areas.

- Feature Properties Area
- Measurement Properties Area
- Advanced Measurement Options Area
- Command Buttons

For additional information, see the "The Auto Feature Dialog Box" topic in the Core PC-DMIS documentation.

Options used for specific auto features are discussed in their respective sections.

Feature Properties Area

XYZ Center or Point - These boxes show the feature’s XYZ center or point location in part coordinates.

IJK Surface, Edge, Slot, or Gap Dir (Vector) - These boxes allow you to set the surface normal vector, edge vector, slot vector, or gap direction of the feature.

IJK Angle Vector - These boxes allow you to define the feature's secondary vector. This helps control the orientation of the feature.

Polar/Cartesian Toggle - This button toggles the display between polar and Cartesian modes.

Find Nearest CAD - When you select an axis (X,Y, or Z) from one of the Center boxes and click this button, PC-DMIS finds the closest CAD element in the Graphic Display window to that axis.

Point Read From Machine - When you click this button, PC-DMIS uses the XYZ location of the machine for the feature’s XYZ coordinates.

Find Vector - This button pierces all surfaces along the XYZ point and IJK vector looking for the closest point. The surface normal vector is displayed as the IJK NOM VEC but the XYZ values do not change.

> **NOTE:** This option is only available for Surface and Edge Point features.

Flip Vector - This button flips the surface normal vector. For example, 0,0,1 would flip to 0,0,-1.

Use Thickness - This button applies a thickness to a feature. When this button is selected you can then specify whether to use actual or theoretical values and provide the value for the thickness.

Swap Vectors - Clicking this button causes the current edge vector and surface vector to switch vectors with each other. **Note:** This option is only available for Edge Point features.

Measure Now - This toggle button determines whether or not PC-DMIS measures the feature when you click **Create**.

Re-Measure - This toggle button determines whether or not PC-DMIS automatically re-measures the feature a second time once the feature has been measured. It will use the measure values from the first measurement as the target locations for the second measurement.

Measurement Properties Area

For information about the specific parameters that are configured in this section, see the following topics:

- Edge Point Specific Parameters
- Plane Specific Parameters
- Circle Specific Parameters
- Slot Specific Parameters
- Flush and Gap Specific Parameters
- Cylinder Specific Parameters
- Sphere Specific Parameters

Auto Wrist - This toggle button will cause the probe orientation to move to a vector that closely corresponds to the surface vector of the Auto Feature.

View Normal - Clicking this button orients that CAD so that you look down on the feature.

View Perpendicular - Clicking this button orients the CAD so that you look at the side of the feature.

**Probe Toolbox Toggle** - Shows/hides the **Probe Toolbox** with the settings for the feature represented in the **Auto Feature** dialog box.

Advanced Measurement Options Area

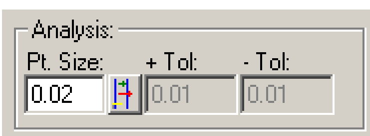
Best Fit Math Type

A Laser Auto Feature Circle also allows you to define the Best Fit Math Type. This is discussed in the "Best Fit Type" topic of the core PC-DMIS documentation. Valid options for the Perceptron system are Maximum Inscribed, Minimum Circumscribed and Least Squares.

Relative to

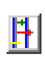
This allows you to keep the relative position and orientation between a given feature (or features) and the auto feature. Click the  button to open the **Relative Feature** dialog box to select the feature or features to which the auto feature is relative. Multiple features can be defined for each axis (XYZ) relative to your auto feature.

Analysis Area



The **Analysis** area allows you to determine how each measured hit/point is displayed.

Pt. Size - Determines how big the measured points are drawn in the **CAD View** tab. This value specifies the diameter, as is specified in current units (mm or inch).

Graphic Analysis button  - When switched on, PC-DMIS does a tolerance check on each point (how far from the calculated actual feature it is), and draw it in the appropriate color, based on the current defined dimension color range.

+ Tol - This provides the positive tolerance from the nominal, and is specified in current measurement routine units. Points that are greater than this value from the nominal are colored based on the standard PC-DMIS plus tolerance color. See the "Editing Dimension Colors" topic in the PC-DMIS Core documentation.

- Tol - This provides the negative tolerance from the nominal, and is specified in current measurement routine units. Points that are less than this value from the nominal are colored based on the standard PC-DMIS negative tolerance color. See the "Editing Dimension Colors" topic in the PC-DMIS Core documentation.

Command Buttons

>> - This button expands the **Auto Feature** dialog box to show additional, more advanced, auto feature options.

<< - This button hides the more complex features of the **Auto Feature** dialog box.

Move To - This button moves the field of view of the Graphic Display window and centers it on the features XYZ location. If the feature is comprised of more than on point (such as a line), then clicking this button switches between the points making up the feature. For a laser slot auto feature the field of view moves to the center of the slot feature.

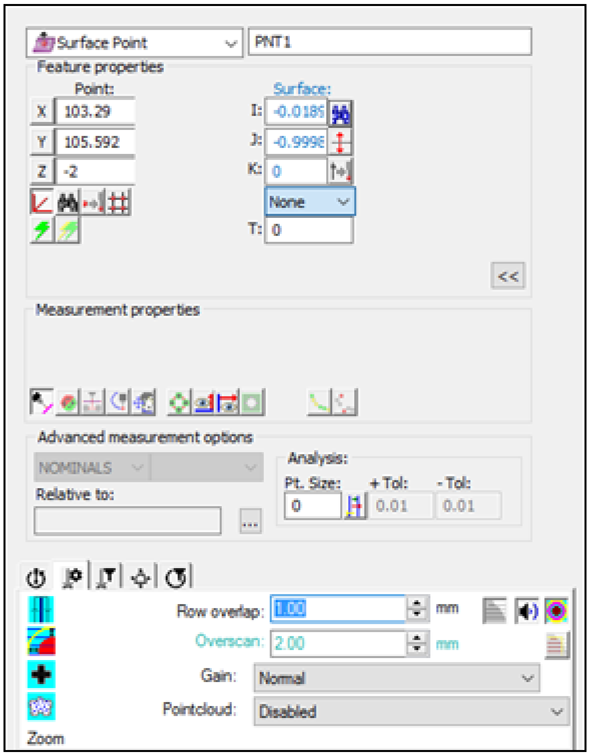
Test - This button tests the auto feature before PC-DMIS creates it. For laser features, the machine will scan over the feature and calculate the measured value for the feature.

Create - This button creates the auto feature and the **Auto Feature** dialog box remains open.

Close - This button closes the **Auto Feature** dialog box without creating a feature.

Laser Surface Point

Three methods exist to calculate the laser surface point: Planar, Spherical and Extended Surface Point. For more information on these methods, see Calculation Methods.



Surface Point auto feature

To measure a laser surface point with a laser sensor:

1. Access the **Auto Feature** dialog box, and click **Surface Point**.
2. Do one of the following:

    - Click on the CAD to give the point a location and vector. Then manually enter any remaining information.
    - From the Graphic Display window, use the **Laser View** tab to move the machine to the point location. Next, from the **Feature Properties** area, click the **Read Point from Position** button. Then manually enter the any remaining information.
    - Manually enter the theoretical information for X, Y, Z, I, J, K, and so on.

3. Enter the necessary information in the **Probe Toolbox** tabs. Cycle through the **Laser Scan Properties**, **Laser Filtering Properties**, and **Laser Clipping Region Properties** tabs to enter the information.
4. If desired, click the **Test** button to test the feature.

WARNING: When you click Test, the machine moves. To avoid injury, stay clear of the laser and the machine. To avoid hardware damage, run the machine at a slower speed.

5. Click **Create** and then **Close**.

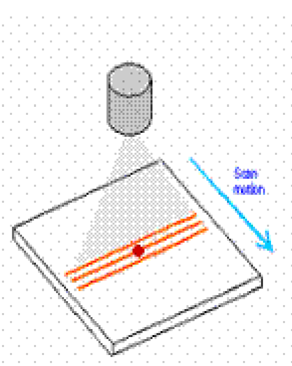
Surface Point Command Mode Text

The Surface Point command inside the Edit window's Command Mode looks like this:

```
PNT1 =FEAT/LASER/SURFACE POINT,CARTESIAN
    THEO/<1.895,1.91,1>,<0,0,1>
    ACTL/<1.895,1.91,1>,<0,0,1>
    TARG/<1.895,1.91,1>,<0,0,1>
    SHOW FEATURE PARAMETERS=YES
        SURFACE=THEO_THICKNESS,1
        MEASURE MODE=NOMINALS
        RMEAS=NONE,NONE,NONE
        AUTO WRIST=NO
        GRAPHICAL ANALYSIS=NO
        FEATURE LOCATOR=NO,NO,""
    SHOW_LASER_PARAMETERS=YES
        POINT CLOUD ID=DISABLED
        SENSOR FREQUENCY=25,OVERSCAN=2,EXPOSURE=18
        FILTER=NONE
```

Auto Surface Point Path

The direction of the path will be determined based on the stripe.



Path direction of scan for surface point

Calculation Methods

There are three methods for calculating laser surface points:

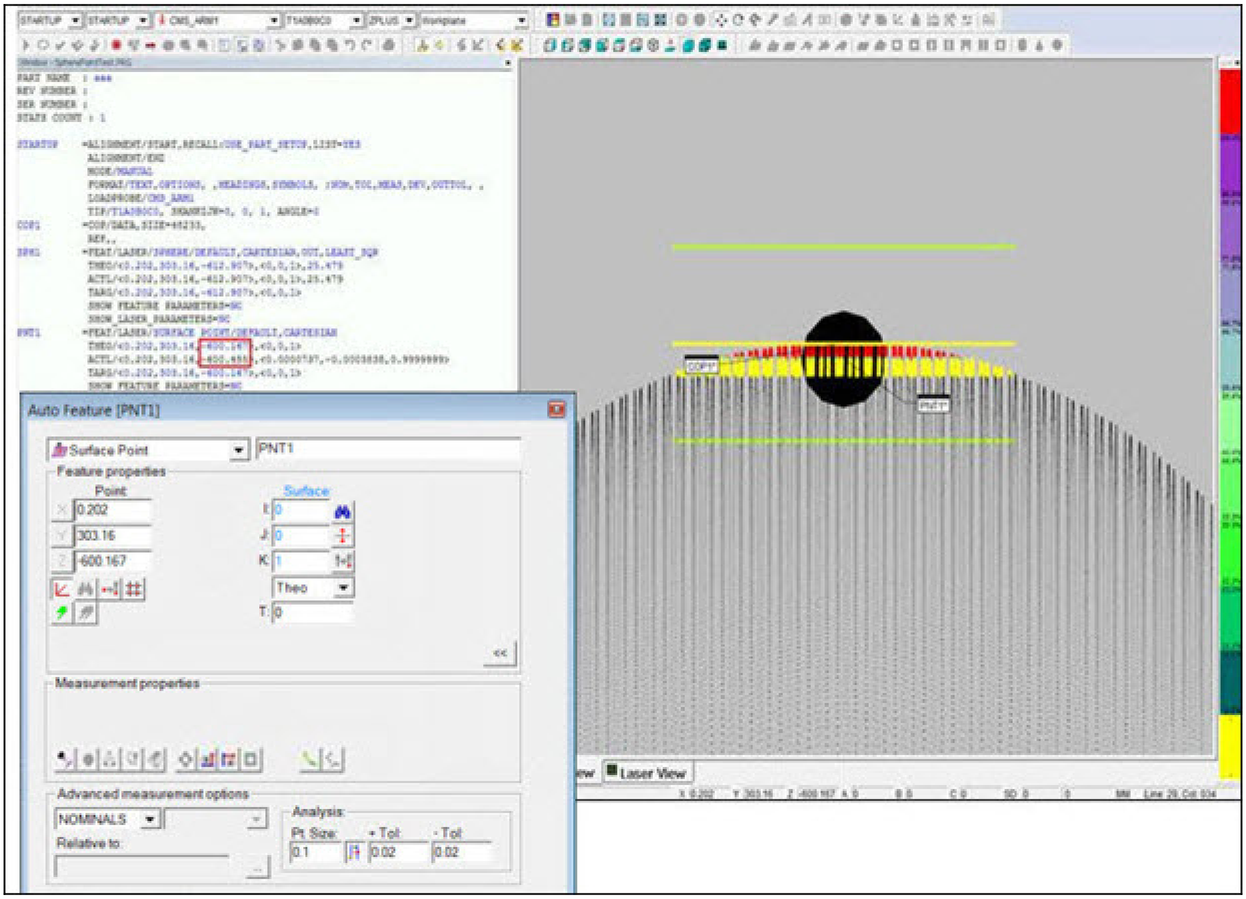
- Planar
- Spherical
- Extended Surface Point

Changing the Calculation Method

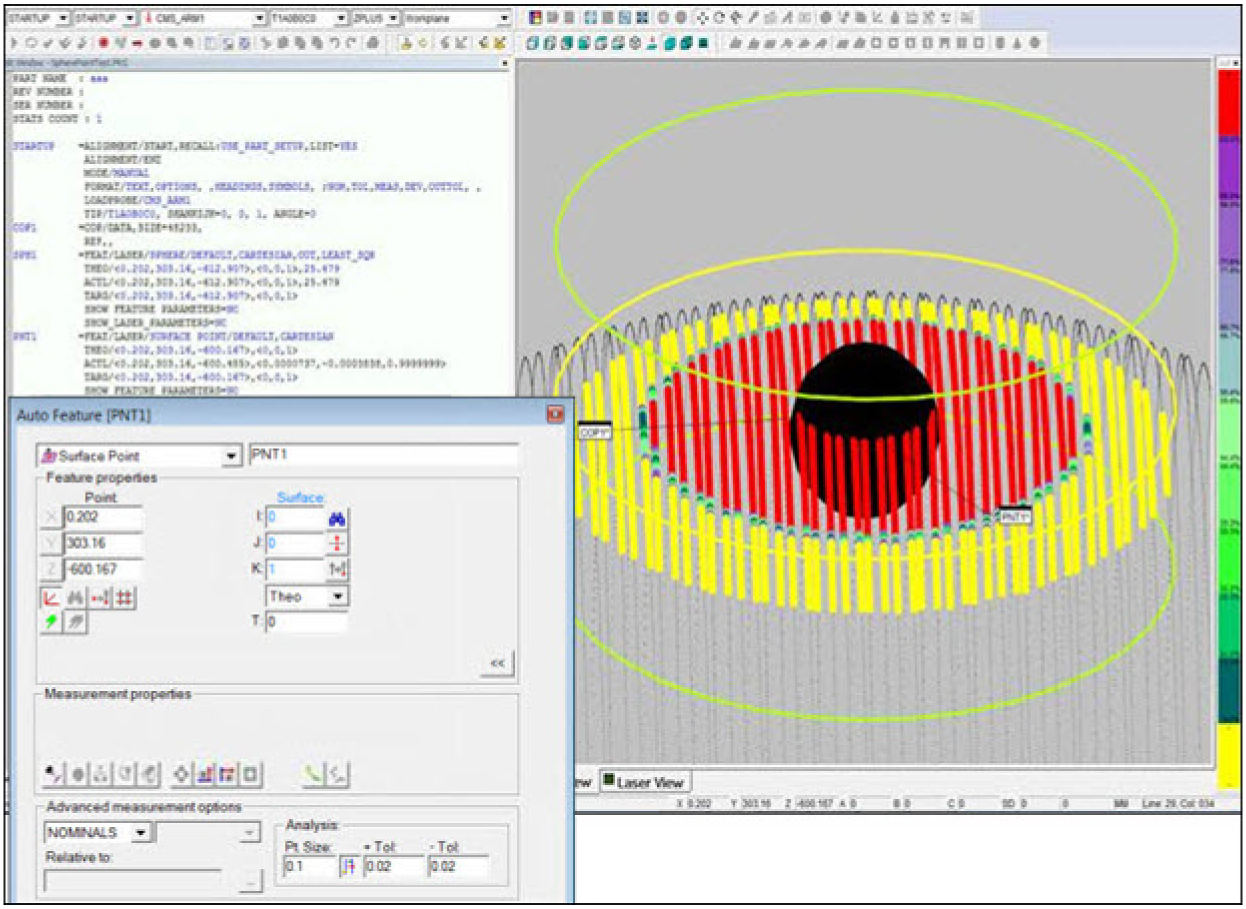
To change the calculation method, modify the `SurfacePointType` registry entry located in the **AutoFeatures** section of the PC-DMIS Settings Editor. For information on this entry, launch your PC-DMIS Settings Editor and press F1 to access its help file. See the PC-DMIS Settings Editor documentation for more information.

Planar Surface Point Calculation Method

This method calculates the laser surface point by fitting a local plane on the scan points within the circular area defined by the horizontal and vertical clipping parameters; this is the default method. Following is an example and its details:



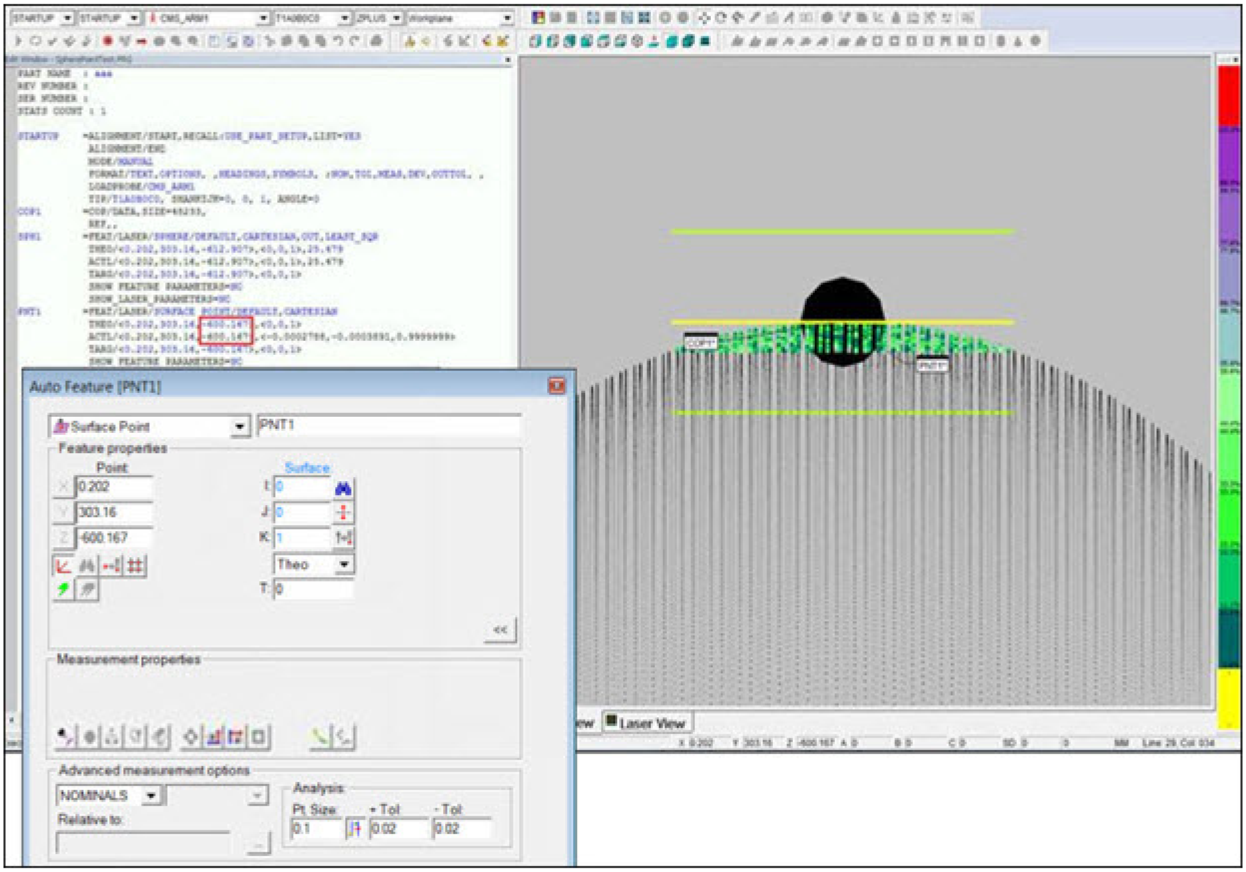
Planar surface point example



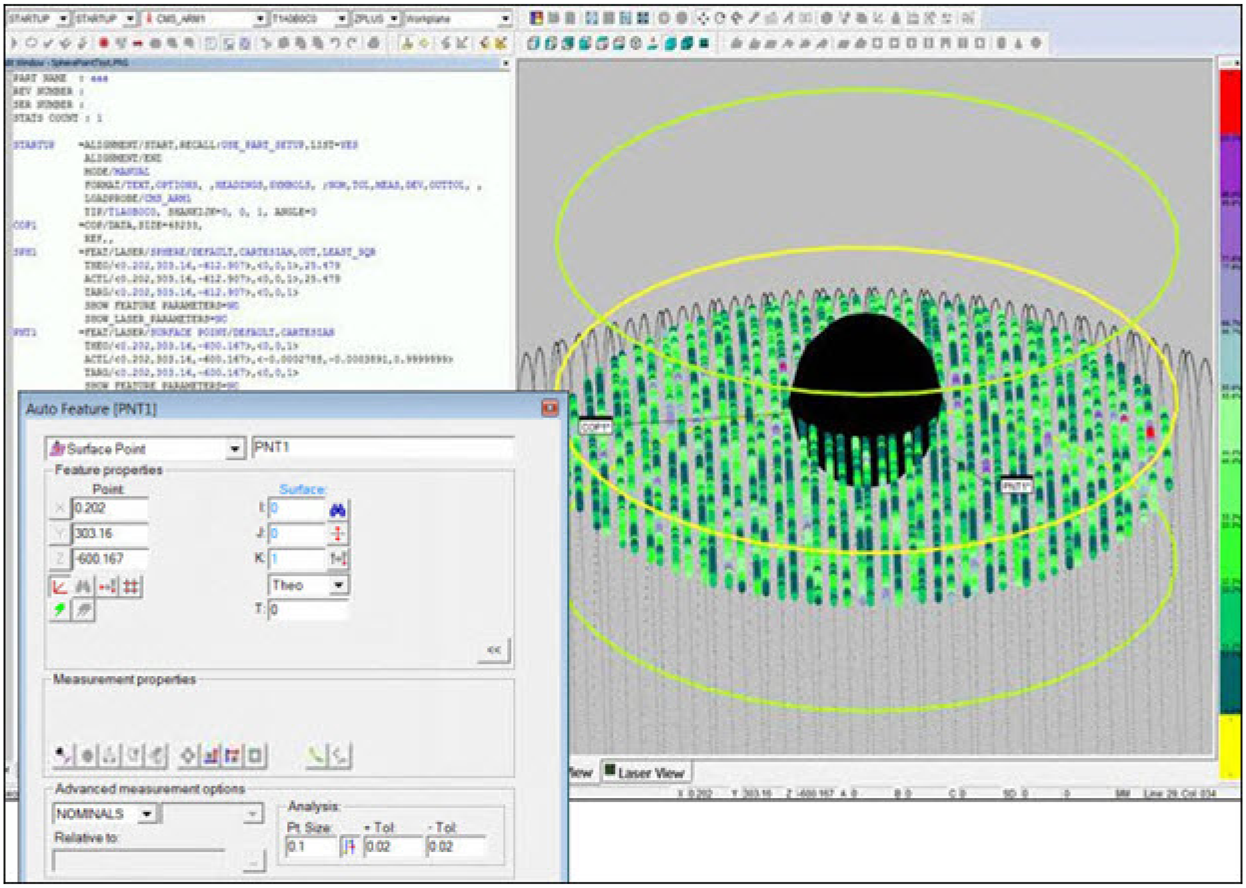
Planar surface point example - details

Spherical Surface Point Calculation Method

This method calculates the laser surface point by fitting a local sphere on the scan points within the circular area defined by the horizontal and vertical clipping parameters. Following is an example and its details:



Spherical surface point example



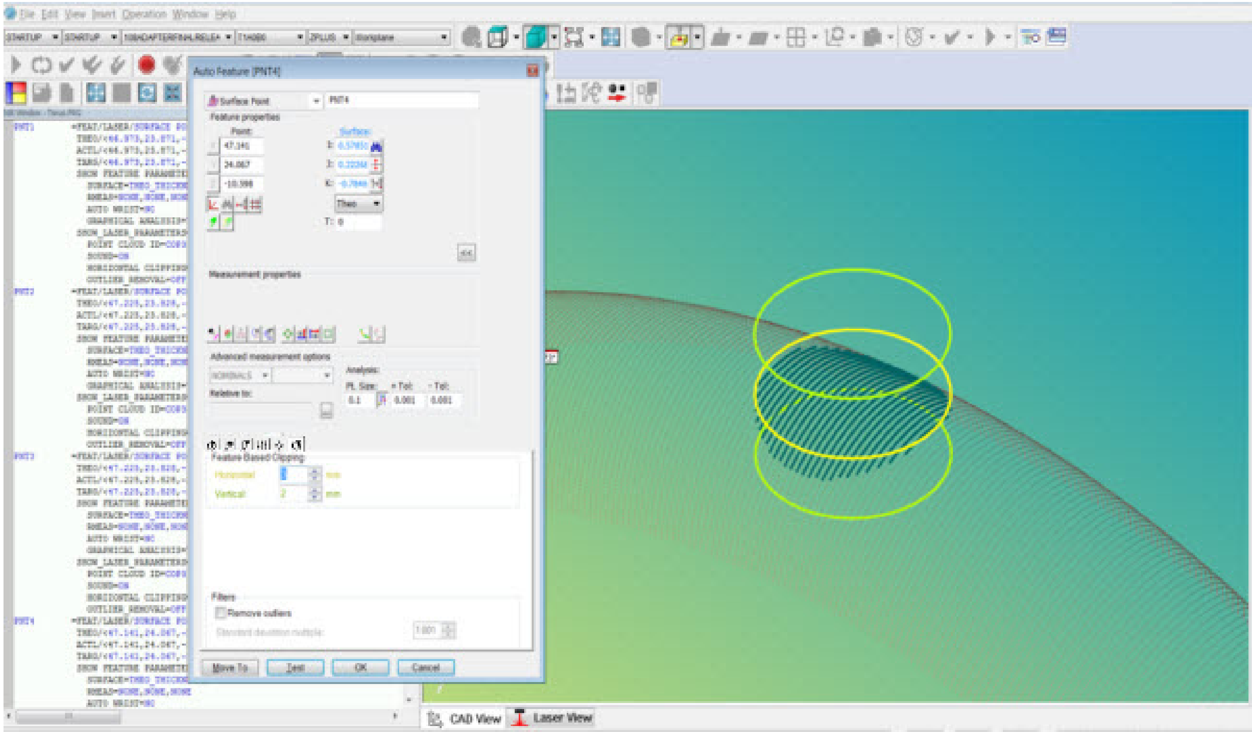
Spherical surface point example - details

Extended Surface Point Calculation Method

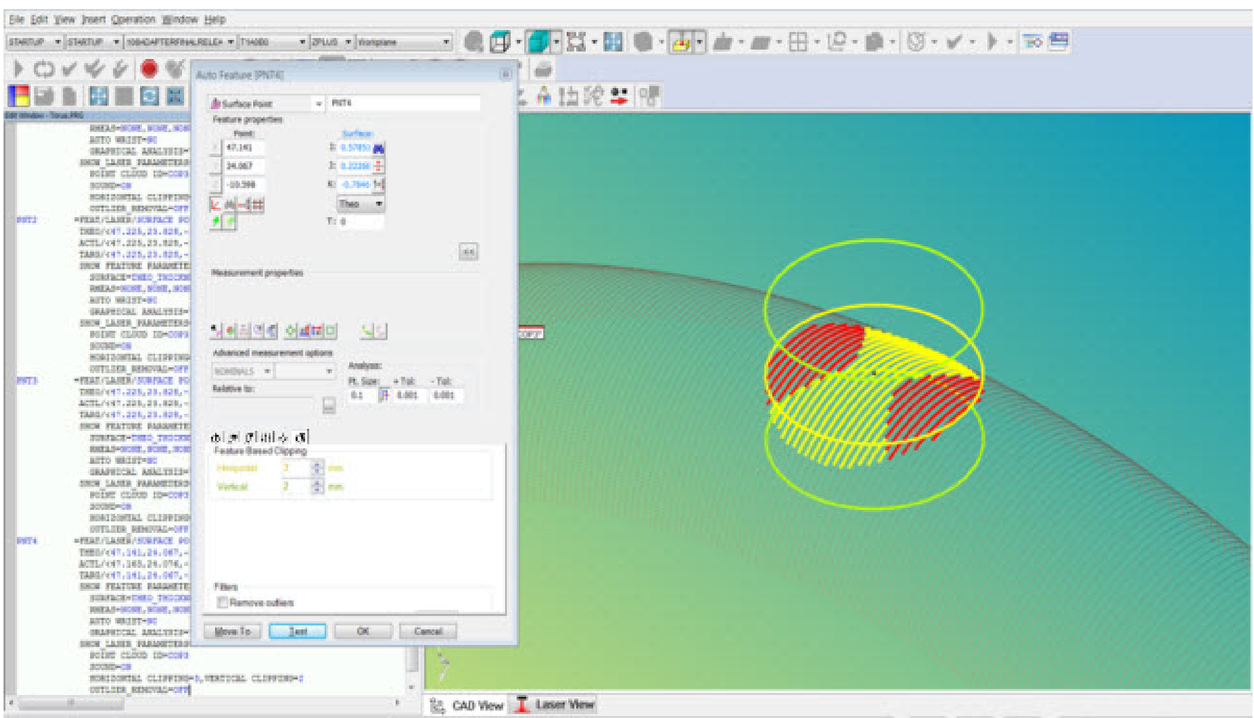
This algorithm is able to calculate the Surface Point by fitting a local 2-curvatures manifold on the scan points within the circular area defined by the horizontal and vertical clipping parameters.

This method is especially useful to calculate Surface Points on fillet surfaces.

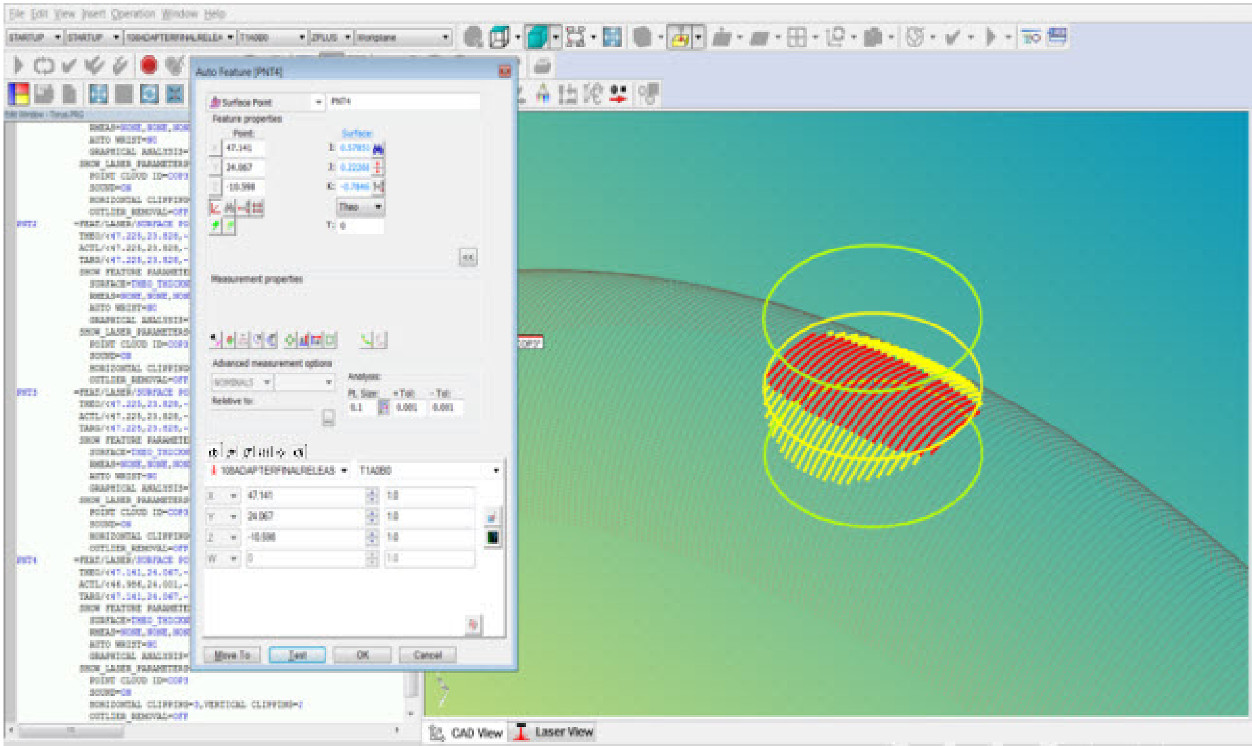
The pictures below show the comparative results of the Extended Surface Point, Extended Spherical Surface Point and Extended algorithms applied to a point on a 2-curvatures filleted surface:



Extended Surface Point details



Extended Spherical Surface Point details



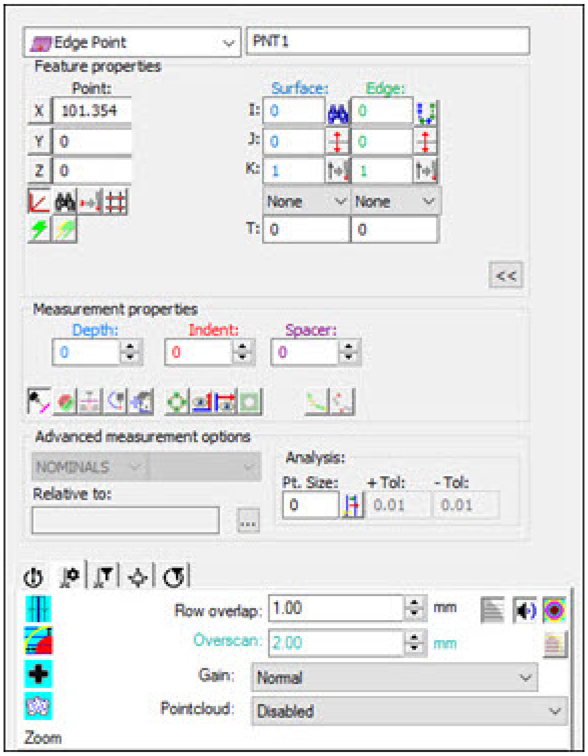
Extended Planar Surface Point details

If the Log file is enabled, additional results from the calculation of Extended Surface Points are available in the file "WaiFE_Debug.txt" found in the C:\ProgramData\Hexagon\PC-DMIS\(PC-DMIS version)\NCSensorsLogs\FeatureExtractor folder:

```
--------------- SURFACE POINT - begin: --------------

                TYPE: EXTENDED
                ACTUAL LOCAL CURVATURES: -0.028572 :  -0.200001
                ACTUAL SURFACE POINT:              i= 47.141291, j= 24.067065, k= -10.597570
                ACTUAL SURFACE VECTOR:             i= 0.553249557, j= 0.232507664, k= -0.799909441
                ACTUAL PRINCIPAL CURVATURE VECTOR: i= -0.832996099, j= 0.147852741, k= -0.533157637
                ACTUAL SECONDARY CURVATURE VECTOR: i= -0.005694434, j= 0.961290671, k= 0.275477440
                STANDARD DEVIATION: 0.000001
                CONDITION INDICATOR: 0.810149

--------------- SURFACE POINT - end --------------
```

The Condition Indicator value is a number from 0 (zero) to 1 inclusive, indicating the quality of the points distribution. 0 (zero) indicates a poor distribution and 1 indicates a good distribution. Generally, a value greater than 0.4 is considered acceptable.

Laser Edge Point



Edge Point Auto Feature dialog box

To measure an edge point with a laser sensor:

1. Access the **Auto Features** dialog box, and select **Edge Point**.
2. Do one of the following:

   - Perform clicks on the CAD to give the point a location and vector. Then manually enter any remaining information.
   - From the Graphic Display window, use the **Laser View** tab to move the machine to the point location. Next, from the **Feature Properties** area, click the **Read Point from Position** button. Then manually enter any remaining information.
   - Manually enter all of the theoretical information for X, Y, Z, I, J, K, and so on.

3. From the **Contact Path Properties** tab of the **Probe Toolbox**, specify values for **Depth**, **Indent**, and **Spacer**. PC-DMIS displays a corresponding graphical visualization of the change in the Graphic Display window.
4. Enter the necessary information in the other **Probe Toolbox** tabs. Cycle through the **Laser Scan**, **Laser Filtering**, and **Laser Clipping**, properties tabs to enter the information.

5. If desired, click the **Test** button to test the feature.

WARNING: When you click Test, the machine moves. To avoid injury, stay clear of the laser and the machine. To avoid hardware damage, run the machine at a slower speed.

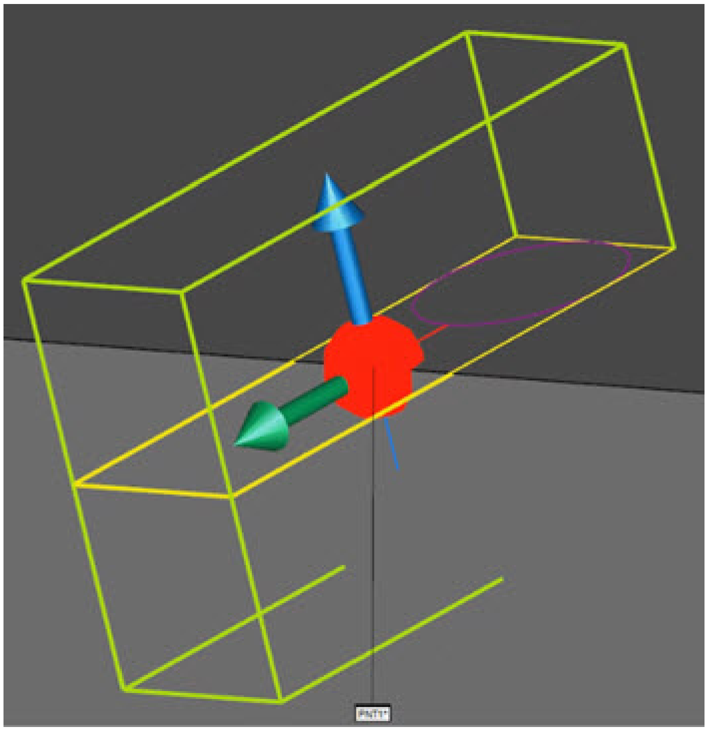
6. Click **Create** and then **Close**.

Edge Point Specific Parameters

Depth: This defines the depth to use when calculating the edge point. This corresponds to the blue graphical visualization in the Graphic Display window. A depth of 0 will cause this feature to be calculated at the surface plane height, using data found at the lowest possible depth from the surface plane. A depth of any other value will cause it to be calculated at that depth.

Spacer: This controls the size of the area PC-DMIS uses to calculate the feature normal. This corresponds to the purple graphical visualization in the Graphic Display window.

Indent: This lets you define the location of the area PC-DMIS uses to calculate the feature normal. This corresponds to the red graphical visualization in the Graphic Display window.



Sample Edge Point with Depth, Spacer, and Indent graphical visualizations used in the Graphic Display window

Notes on Graphical Analysis and Feature Extraction of Edge Points

If you don't see some graphical analysis points computed to the edge plane, consider the following:

- **Edge Line Points** - All the edge line points on the reference plane returned by the feature extractor are displayed. For analysis, the edge line points are computed using the distance (**Indent** value) from the reference plane center (center of the circular surface area defined by the **Spacer** value) to the edge line.
- **Reference Plane Points** - If the Spacer value is 0.0 then the reference plane points are not displayed. If the Space value is not 0.0 then the reference plane points are extracted from the point cloud, applying the following rules using the plane statistical data returned by the feature extractor:

  - Rule 1: All points that are outside of an *imaginary cylinder* are discarded.

    This cylinder is identified by using the following values:

    Center = Indent Center Point

    Vector = Surface Vector

Radius = Spacer

- Rule 2: All points with a distance from an *imaginary plane* greater than the maximum plane error value are discarded.

  This plane is identified by using the following values:

  Center = Measured Edge Point

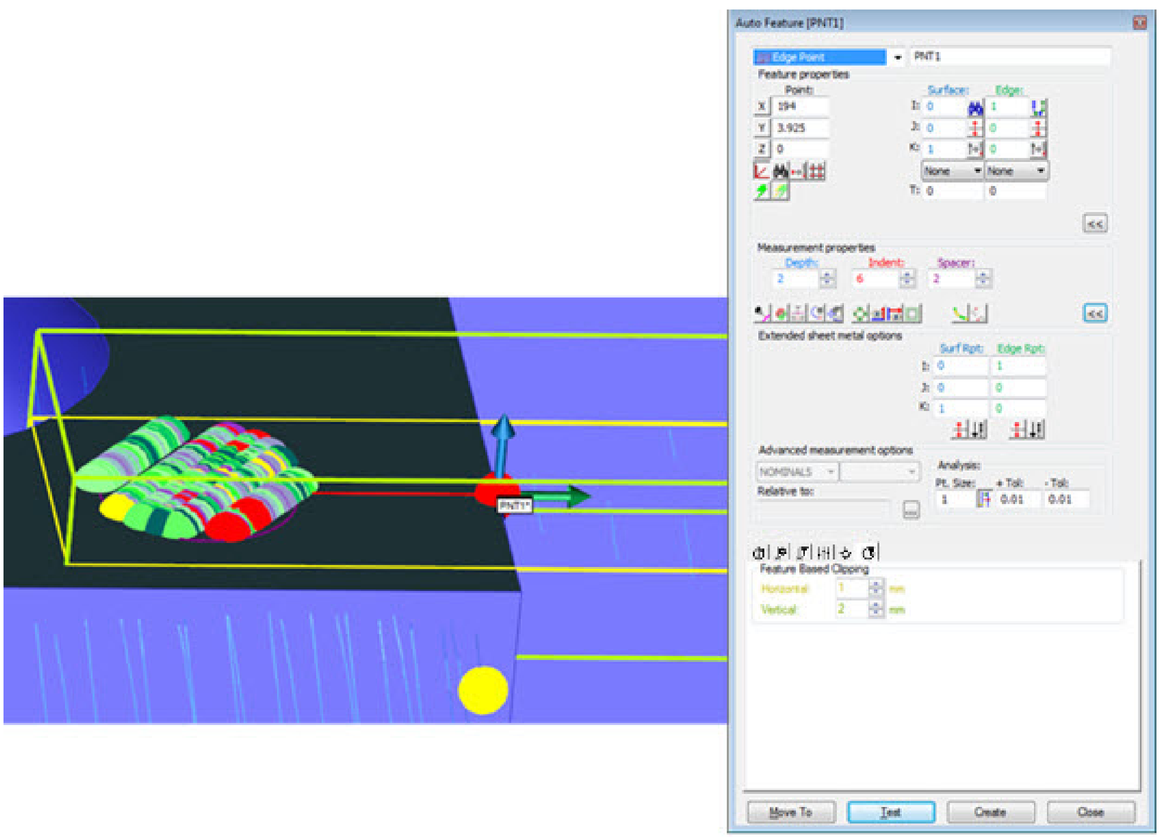
  Vector = Measured Surface Vector

- Rule 3: If any remaining points are higher than the allowed number (19900), the points are uniformly reduced to the allowed value.

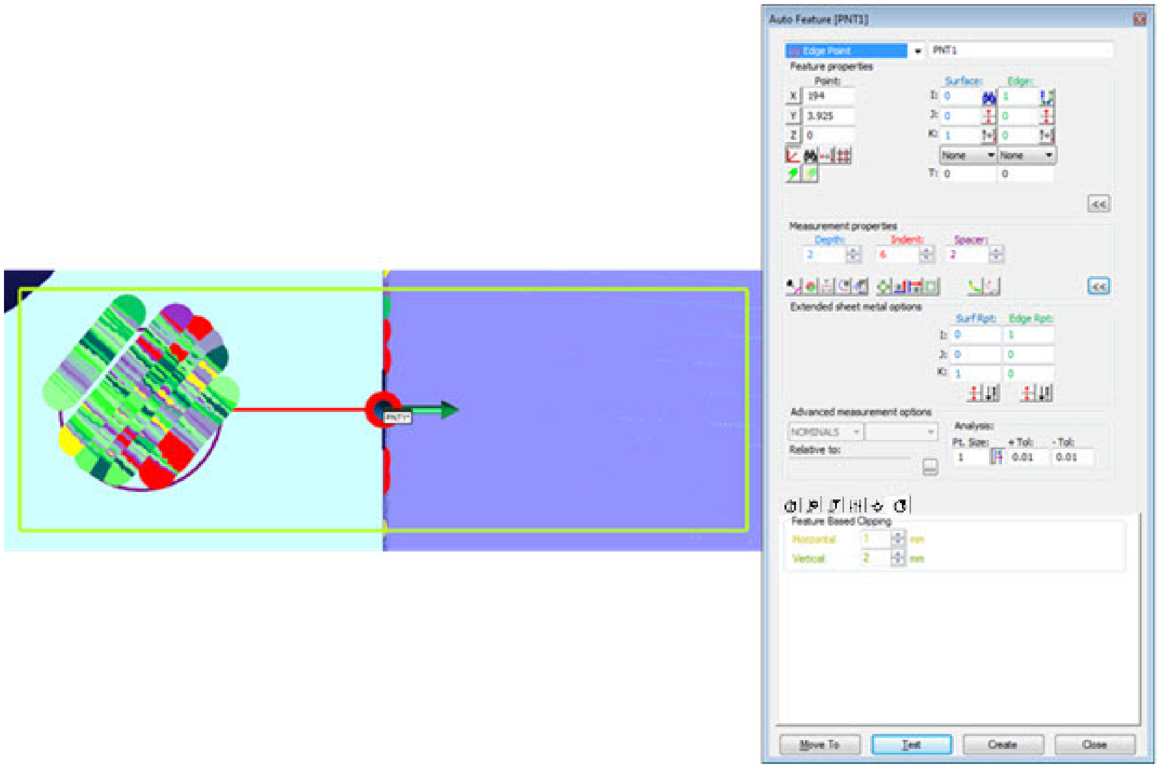
For analysis, each reference plane point is computed using the distance from the reference plane and the measured surface plane.

The following two images show the Edge Point laser graphical analysis:

- *Graphical Analysis Example - Side View*



- *Graphical Analysis Example - Top View*



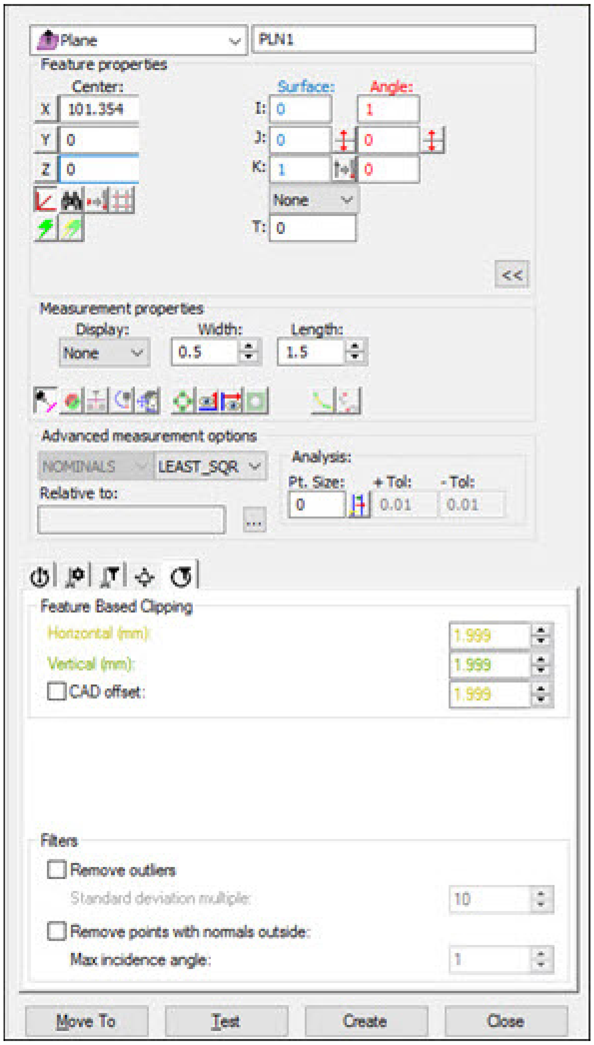
Edge Point Command Mode Text

The Edge Point command inside the Edit window's Command Mode looks like this:

```
PNT2 =FEAT/LASER/EDGE POINT,CARTESIAN
    THEO/<1.895,1.91,1>,<0,1,0>,<0,0,1>
    ACTL/<1.895,1.91,1>,<0,1,0>,<0,0,1>
    TARG/<1.895,1.91,1>,<0,1,0>,<0,0,1>
    SHOW FEATURE PARAMETERS=YES
        SURFACE1=THEO_THICKNESS,1
        SURFACE2=THEO_THICKNESS,0
        MEASURE MODE=NOMINALS
        RMEAS=NONE,NONE,NONE
        AUTO WRIST=NO
        GRAPHICAL ANALYSIS=NO
        FEATURE LOCATOR=NO,NO,""
    SHOW_LASER_PARAMETERS=YES
        POINT CLOUD ID=DISABLED
```

```
SENSOR FREQUENCY=25,OVERSCAN=2,EXPOSURE=18
FILTER=NONE
```

Laser Plane



Plane Auto Feature dialog box

To create an auto plane using a laser sensor:

1. Access the **Auto Features** dialog box (**Insert | Feature | Auto**), and select **Plane**.
2. Do one of the following:

    - Click on the CAD to give the plane a location and vector. Then manually enter any remaining information.

- From the Graphic Display window, use the **Laser View** tab to move the machine to the center of the plane location. Click the **Read Point from Position** button. Manually enter the any remaining information like the display, width, length, and so on.
- Manually enter the theoretical information for theoretical X, Y, Z, I, J, K, display, width, length, and so on.

3. Enter the necessary information in the **Probe Toolbox** tabs. Cycle through the **Laser Scan**, **Laser Filtering**, and **Laser Clipping**, properties tabs to enter the information as needed.
4. If desired, click the **Test** button to test the feature.

WARNING: When you click Test, the machine moves. To avoid injury, stay clear of the laser and the machine. To avoid hardware damage, run the machine at a slower speed.

5. Click **Create** and then **Close**.

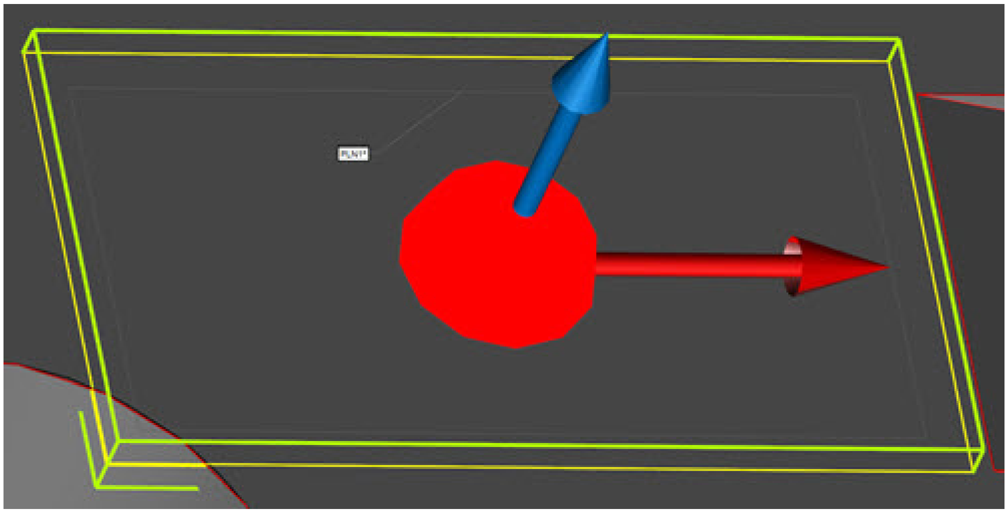
Plane Specific Parameters

Width: The value in this box determines the width of the measurement area of the plane.

Length: The value in this box determines the length of the measurement area of the plane.

Display: This list lets you choose how to present the plane inside the Graphic Display window. You can choose **NONE** or **TRIANGLE** or **OUTLINE**:

- If you choose **NONE**, the plane is not displayed.
- If you choose **TRIANGLE**, PC-DMIS displays the plane with a triangle symbol at the very center of the plane.
- If you choose **OUTLINE**, PC-DMIS displays an outline of the edges of the plane.



Sample Plane in the Graphic Display window with:

***Outline** display (Gray dotted line)*

***Overscan** display (Yellow rectangle)*

***Vertical Clipping** (Green rectangular box)*

Plane Command Mode Text

The Plane command inside the Edit window's Command Mode looks like this:

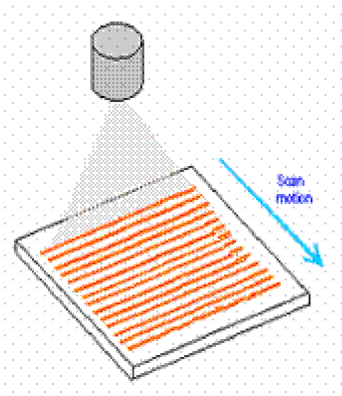
```
PNT1 =FEAT/LASER/EDGE POINT/DEFAULT,CARTESIAN,TRIANGLE
    THEO/<-19.594,3.822,0>,<-1,0,0>,<0,0,1>
    ACTL/<-19.594,3.822,0>,<-1,0,0>,<0,0,1>
    TARG/<-19.594,3.822,0>,<-1,0,0>,<0,0,1>
    DEPTH=4
    INDENT=7
    SPACER=1
    SHOW FEATURE PARAMETERS=YES
        SURFACE1=THEO_THICKNESS,0
        SURFACE2=THEO_THICKNESS,0
        RMEAS=NONE,NONE,NONE
        AUTO WRIST=NO
        GRAPHICAL ANALYSIS=NO
    SHOW_LASER_PARAMETERS=YES
```

```
POINT CLOUD ID=COP2
HORIZONTAL CLIPPING=9,VERTICAL CLIPPING=9
```

Auto Plane Paths

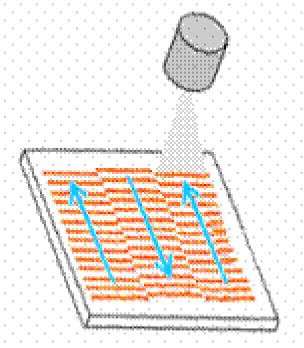
PC-DMIS provides two different paths for a plane. It automatically chooses the appropriate path based on the diameter and the size of the usable portion of the laser stripe. For auto planes, PC-DMIS always scans perpendicular to the direction of the stripe.

Path 1: Smaller Width



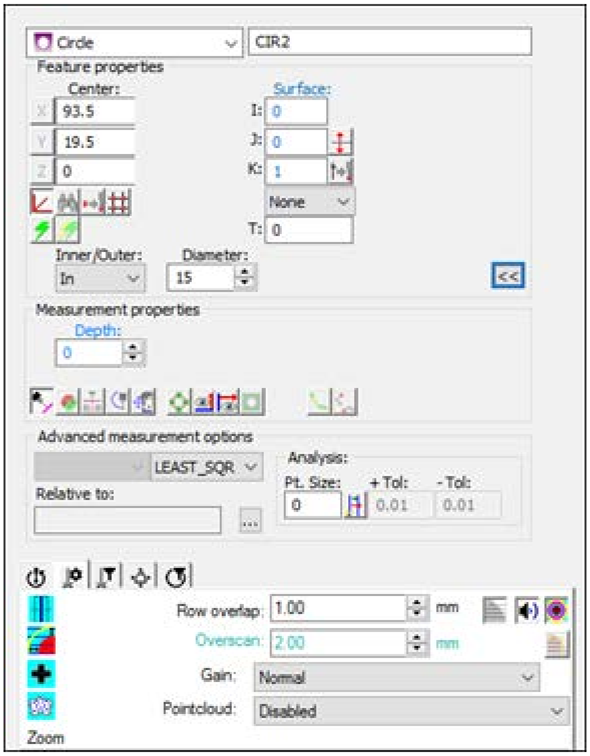
Planes with a width smaller than the usable portion of the stripe

Path 2: Larger Width



Planes with a width larger than the usable portion of the stripe

Laser Circle



Circle Auto Feature

To create a laser auto circle:

1. Access the **Auto Features** dialog box, and select **Circle**.
2. Do one of the following:

    - Perform clicks on the CAD to give the circle a location and vector. Then manually enter any remaining information.
    - From the Graphic Display window, use the **Laser View** tab to move the machine to the circle location. Next, from the **Feature Properties** area, click **Read Point from Machine** . Then manually enter any remaining information, like the diameter, depth, and so on.
    - Manually enter all of the theoretical information for X, Y, Z, I, J, K, diameter, depth, and so on.

3. Enter the necessary information in the **Probe Toolbox** tabs. Cycle through the **Laser Scan Properties**, **Laser Filtering Properties**, and **Laser Clipping Properties** tabs to enter the information.
4. If desired, click the **Test** button to test the feature.

> **WARNING:** When you click Test, the machine moves. To avoid injury, stay clear of the laser and the machine. To avoid hardware damage, run the machine at a slower speed.

5. Click the **Create** button and then click **Close.**

> **NOTE:** Currently, you can only measure inner circles (holes) with laser sensors.

Circle Specific Parameters

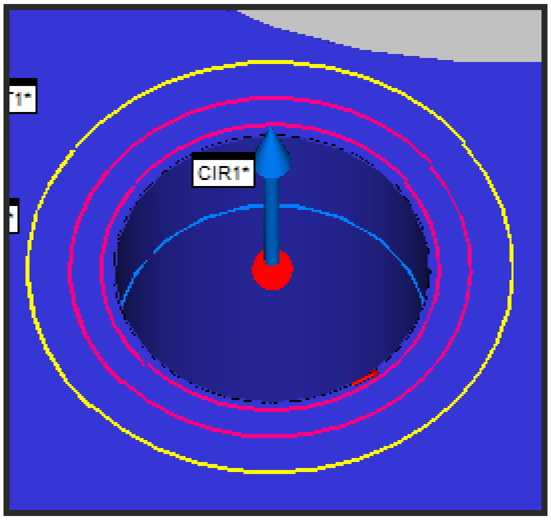
Diameter - This box specifies the circle’s diameter. When you select a circle with the mouse in the Graphic Display window, PC-DMIS automatically places the circle’s diameter from the CAD model in this box.

Depth - This parameter controls what data PC-DMIS uses to calculate the feature characteristics. You can use the depth value to eliminate data on a chamfer or some other transitional portion of the feature that you don't want in the feature calculation. Specifying a positive value tells PC-DMIS where along the feature PC-DMIS looks to calculate the feature characteristics. A depth of 0 will cause this feature to be calculated at the surface plane height, using data found at the lowest possible depth from the surface plane. A depth of any other value causes it to be calculated at that depth. Due to hardware limitations, for this feature type if you do use a depth value higher than 0, you must use a minimum of 0.3 mm (.01181 inches).

> **IMPORTANT:** The depth defaults to zero. This is the default value for a plane feature with no extruded edges. You should only change this to a different value if there are specific requirements from your part's drawing. Otherwise, PC-DMIS unsuccessfully attempts to locate points at the specified depth, resulting in a feature calculation error from the feature extraction module.

For example, a depth of 3 indicates that you want to use all data 3 mm (or inches, depending on the units of the measurement routine) and above for the calculation. If you specify 0, this indicates that you want to use all available data for the calculation. For thin walled features, a 0 value may be valid; but for parts with any depth to them, you will probably need to specify a depth to get accurate results.

> **NOTE:** Even if you specify a depth greater than zero, the measured results are always projected into the plane where the feature resides.



Sample Circle in the Graphic Display window showing:
Depth (blue circle)
Ring Band (pink circles)
Overscan (yellow circle)

Auto Circle Command Mode Text

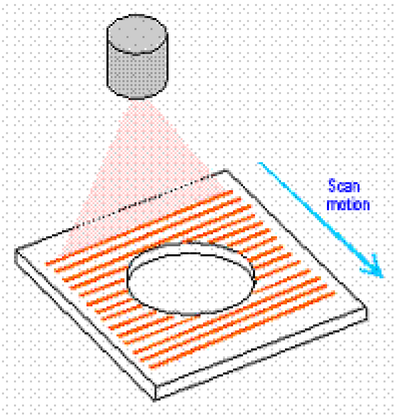
The auto circle command inside the Edit window's Command Mode looks like this:

```
CIR2 =FEAT/LASER/CIRCLE,CARTESIAN
    THEO/<1.895,1.91,1>,<0,0,1>,1.895
    ACTL/<1.895,1.91,1>,<0,0,1>,1.895
    TARG/<1.895,1.91,1>,<0,0,1>
    ANGLE VEC=<0,0,1>
    DEPTH=3
    SHOW FEATURE PARAMETERS=YES
        MEASURE MODE=NOMINALS
        RMEAS=NONE,NONE,NONE
        AUTO WRIST=NO
        GRAPHICAL ANALYSIS=NO
        FEATURE LOCATOR=NO,NO,""
    SHOW_LASER_PARAMETERS=YES
        POINT CLOUD ID=DISABLED
        SENSOR FREQUENCY=25,OVERSCAN=2,EXPOSURE=18
        FILTER=NONE
```

Auto Circle Paths

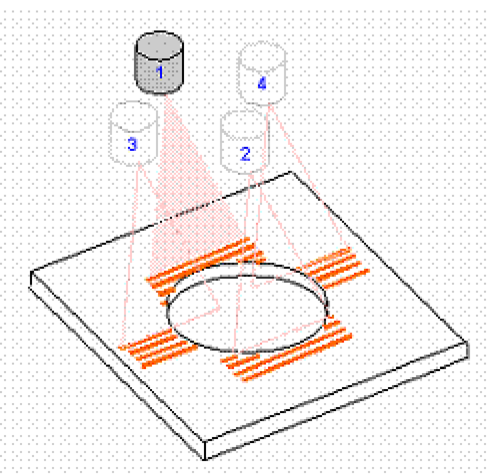
PC-DMIS provides two different paths for a circle. It automatically chooses the appropriate path based on the diameter and the size of the usable portion of the laser stripe. For auto circles, PC-DMIS always scans perpendicular to the direction of the stripe.

Path 1: Smaller Diameter



Circles with a diameter smaller than the usable portion of the stripe

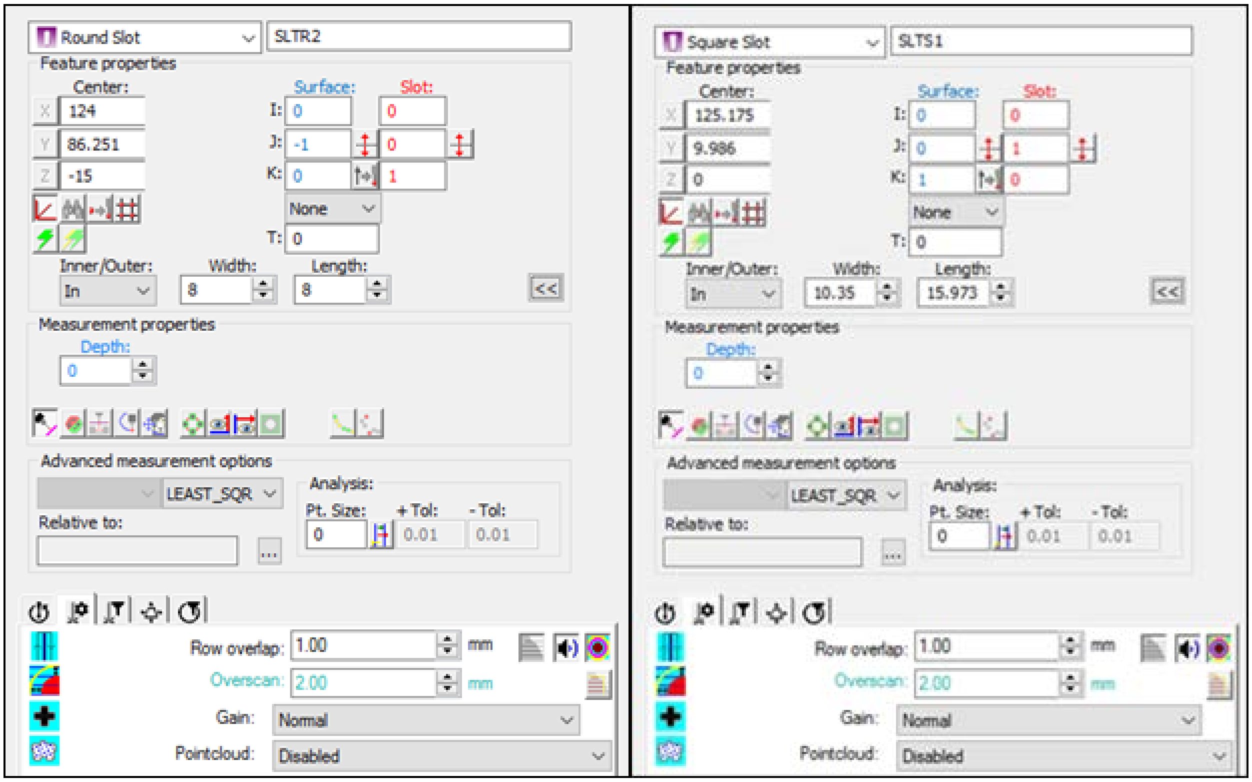
Path 2: Larger Diameter



Circles with a diameter larger than the usable portion of the stripe

NOTE: The method for measuring circles with a larger diameter has been improved to measure the 4 passes at 1:30, 4:30, 7:30, and 10:30 rather than 12:00, 3:00, 6:00, and 9:00 as depicted in the image.

Laser Slot



Auto Feature dialog box - Round Slot left, Square Slot right

To measure a slot with a laser sensor:

1. Access the **Auto Features** dialog box (**Insert | Feature | Auto**), and select **Round Slot** or **Square Slot**.
2. Do one of the following:

   a. Click on the CAD to collect the X, Y, Z, I, J, K information:

   For Round Slots:

   1. Click one of the rounded edges of the slot in the Graphic Display window. PC-DMIS prompts you to click two more times on the same rounded edge.
   2. Click twice on this edge. PC-DMIS then prompts you to click on the other rounded edge.
   3. Click on the other rounded edge. PC-DMIS prompts you to click two more times on that same rounded edge.
   4. Click twice on the second rounded edge. PC-DMIS establishes the orientation of the round slot.

For Square Slots:

1. Click on one of the long edges of the slot in the Graphic Display window. PC-DMIS prompts you to click another location on the same edge to determine the direction.
2. Click on a second edge, 90 degrees from the first.
3. Click on the third edge, 90 degrees from the second. This sets the width.
4. Click on the fourth edge and final edge. This sets the length.

b. From the Graphic Display window, use the **Laser View** tab and move the machine to the slot location. Next, from the **Feature Properties** area, click the **Read Point from Position** button.

3. Manually enter all of the theoretical X, Y, Z, I, J, K, width, length, depth, height, and so on.
4. Enter the necessary information in the **Probe Toolbox** tabs. Cycle through the **Laser Scan**, **Laser Filtering**, and **Laser Clipping**, properties tabs to enter the information.
5. If desired, click the **Test** button to test the feature.

WARNING: When you click Test, the machine moves. To avoid injury, stay clear of the laser and the machine. To avoid hardware damage, run the machine at a slower speed.

6. Click **Create** and then **Close**.

Slot Specific Parameters

Inner/Outer - This list lets you choose whether or not the slot is an Inner slot (a hole) or an Outer slot (a stud).

Width - The value in this box determines the slot’s width.

Length - The value in this box determines the slot’s length.

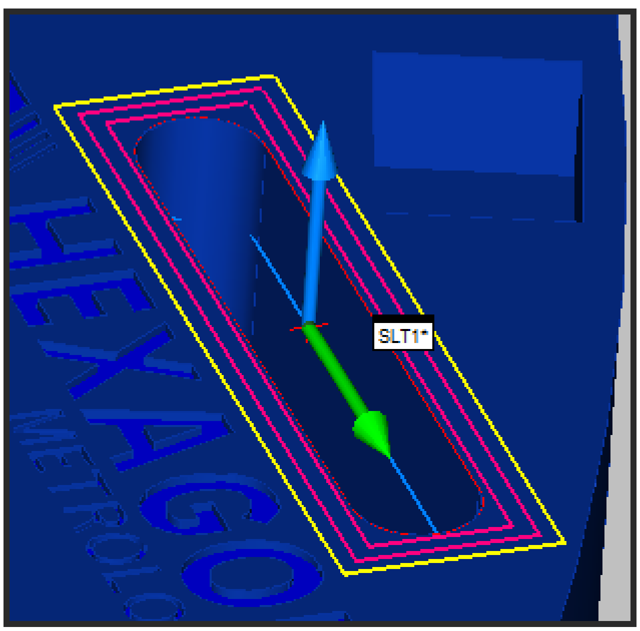
Depth - This parameter controls what data PC-DMIS uses to calculate the feature characteristics. You can use the depth value to eliminate data on a chamfer or some other transitional portion of the feature that you don't want in the feature calculation. A depth of 0 causes this feature to be calculated at the surface plane height, using data found at the lowest possible depth from the surface plane. A depth of any other value causes it to be calculated at that depth. Specifying a positive value tells PC-DMIS where along the feature PC-DMIS looks to calculate the feature characteristics. Due to hardware limitations, for this feature type if you do use a depth value higher than 0, you must use a minimum of 0.3 mm (.01181 inches).

For example, a depth of 3 indicates that you want to use all data 3 mm (or inches, depending on the units of the measurement routine) and above for the calculation. If you specify 0, this indicates that you want to use all available data for the calculation. For thin walled features, a 0 value may be valid; but for parts with any depth to them, you should specify a depth to get accurate results.

NOTE: Even if you specify a depth greater than zero, PC-DMIS always projects the measured results into the plane where the feature resides.

The depth defaults to zero. This is the default value for a plane feature with no extruded edges. You should only change this to a different value if there are specific requirements from your part's drawing. Otherwise, PC-DMIS unsuccessfully attempts to locate points at the specified depth, resulting in a feature calculation error from the feature extraction module.

Slot (Vector) - These boxes define the slot’s orientation.



Sample Round Slot in the Graphic Display window showing:
Depth (blue slot line)
Ring Band (pink rectangles)
Overscan (yellow rectangle)

Slot Command Mode Text

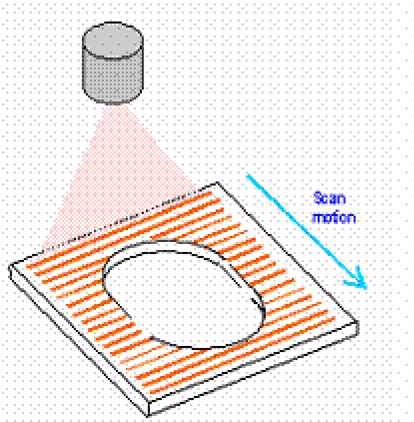
The Slot command inside the Edit window's Command Mode looks like this:

```
SLT1 =FEAT/LASER/SQUARE SLOT,CARTESIAN
    THEO/<1.895,1.91,1>,<0,0,1>,<0,1,0>,3,7
    ACTL/<1.895,1.91,1>,<0,0,1>,<0,1,0>,3,7
    TARG/<1.895,1.91,1>,<0,0,1>
    DEPTH=3
    SHOW FEATURE PARAMETERS=YES
        SURFACE=THEO_THICKNESS,1
        MEASURE MODE=NOMINALS
        RMEAS=NONE,NONE,NONE
        AUTO WRIST=NO
        GRAPHICAL ANALYSIS=NO
        FEATURE LOCATOR=NO,NO,""
    SHOW_LASER_PARAMETERS=YES
        POINT CLOUD ID=DISABLED
        SENSOR FREQUENCY=25,OVERSCAN=2,EXPOSURE=18
        FILTER=NONE
```

Auto Round Slot Paths

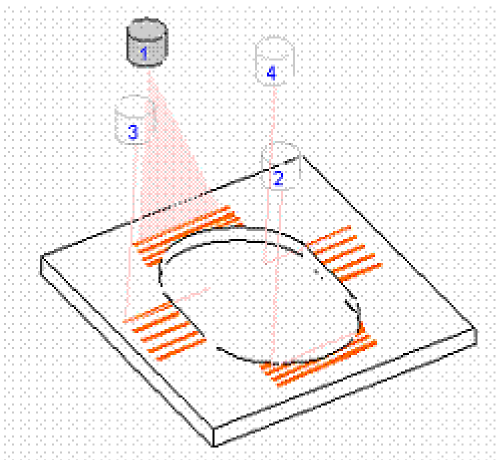
Depending on the width of the round slot, PC-DMIS takes one of these paths when performing the measurement:

Path 1: Narrow Width



Round slots with a width less than the usable portion of the stripe

Path 2: Larger Width

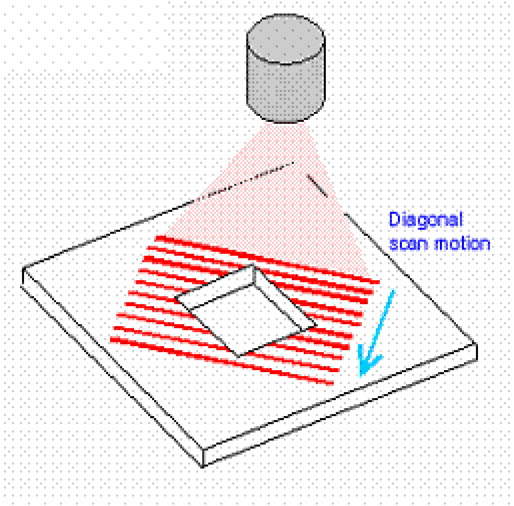


Round slots with a width larger than the usable portion of the stripe

Auto Square Slot Paths

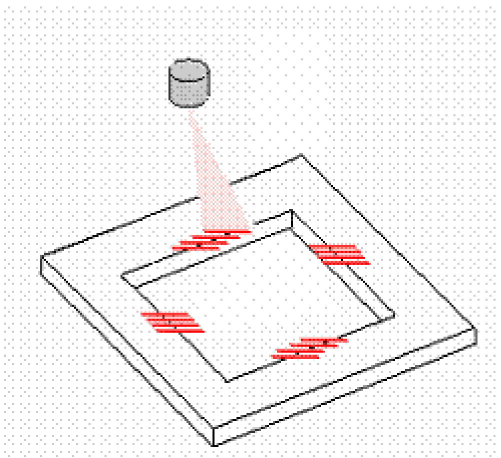
PC-DMIS must measure Auto Square slots at a 45 degree angle to the slot (see the illustrations below). Depending on the size of the slot, PC-DMIS takes one of these two paths.

Path 1: Small Slot - Measured with a single pass of the laser sensor



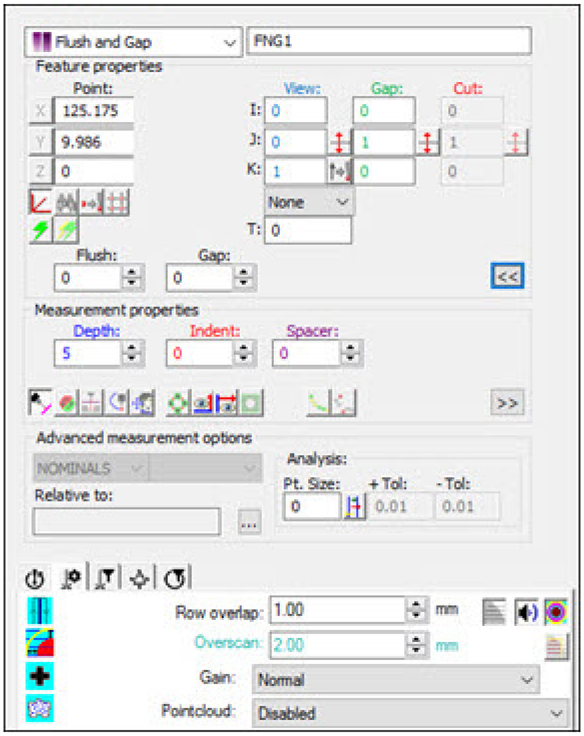
Small square slots require a single pass of the laser sensor's stripe

Path 2: Large Slot - Measured with multiple passes of the laser sensor



Large square slots require multiple passes of the laser sensor's stripe

Laser Flush and Gap



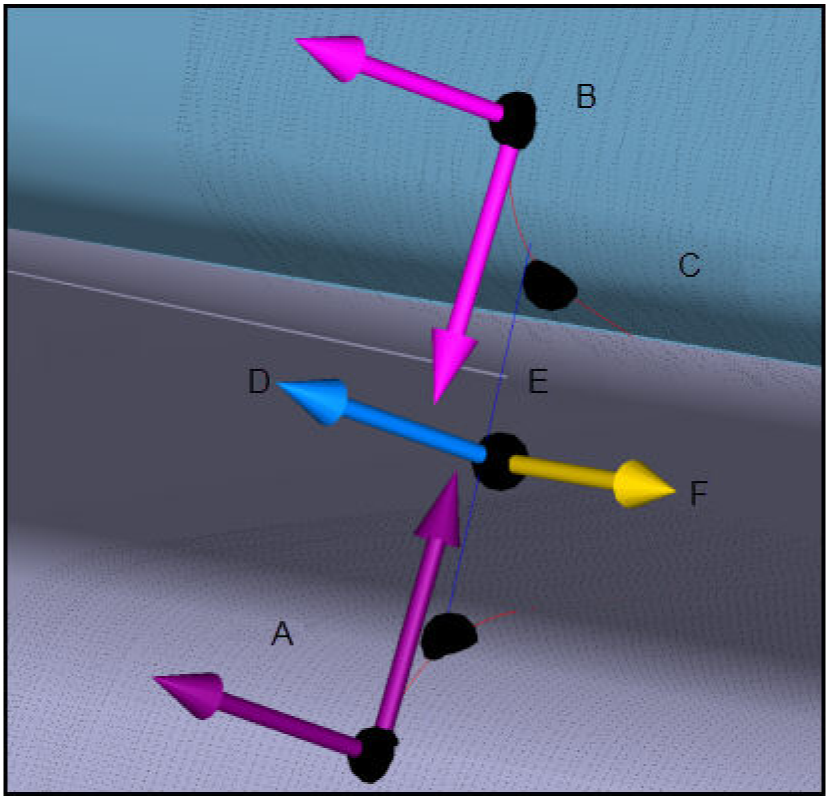
Flush and Gap Auto Feature

Flush and Gap measures the height difference between two mating sheet metal parts (the Flush) and the distance between two mating parts (the Gap).

To measure a Flush and Gap using a laser sensor, access the **Auto Features** dialog box and select **Flush and Gap**. The dialog box automatically expands the **Extended sheet metal options** area. This area provides **XYZ** position boxes and **IJK** vector boxes for the Master and Gauge points. Follow one of the procedures below.

With CAD Data

1. Load a CAD model.
2. Click on the Master side.
3. Click on the Gauge side.



A - Master

B - Gauge

C - CAD learned curves

D - View Vector

E - Depth line

F - Cut Vector

4. These points must be on the reference "flat" surfaces, where PC-DMIS sets the planes used to calculate flush, and not on the curves.
5. PC-DMIS learns the theoretical Flush.
6. PC-DMIS learns the curves from the CAD model.
7. PC-DMIS learns the point coordinate and vectors for both the Master and Gauge sides of the gap.
8. PC-DMIS applies the defined Depth value, and after piercing the curves it calculates the theoretical gap at the specified depth.
9. PC-DMIS also calculate the cut vector (along the rail) and the Gap direction (crossing the rail).
10. Set the **Indent** and the **Spacer** values so that they only include points on the flat surfaces and not points on the curved portion.
11. Set other parameters as needed. See "Flush and Gap Specific Parameters".

12. Enter the necessary information in the **Probe Toolbox** tabs. Cycle through the **Laser Scan**, **Laser Filtering**, and **Laser Clipping**, properties tabs to enter the information.
13. If desired, click the **Test** button to test the feature.

> **WARNING:** When you click Test, the machine moves. To avoid injury, stay clear of the laser and the machine. To avoid hardware damage, run the machine at a slower speed.

14. Click the **Create** button and then **Close**.

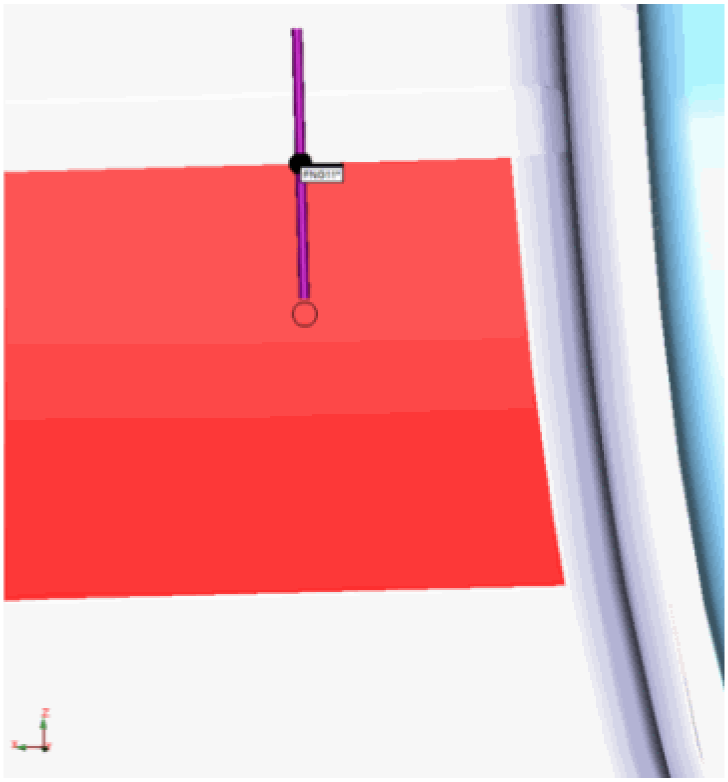
Flush and Gap CAD Selection Capability

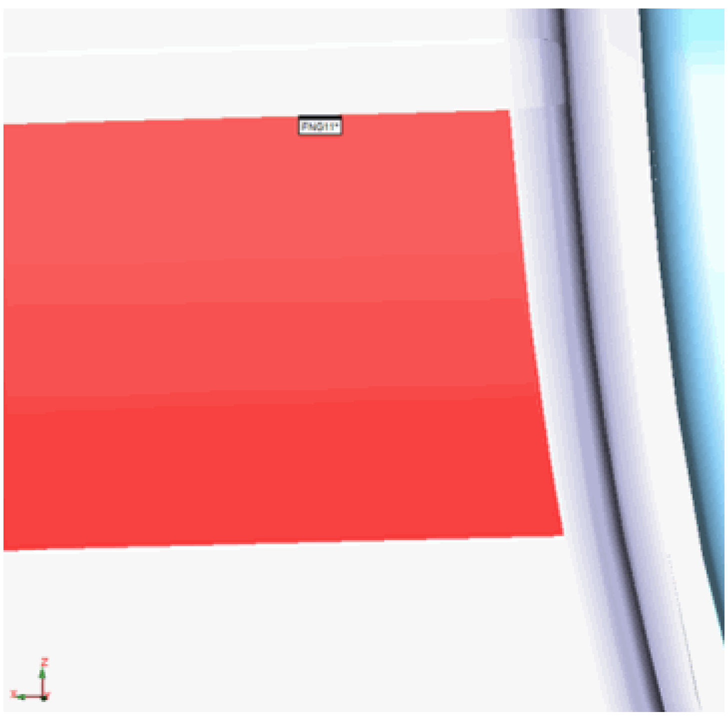
The ability to re-click the first CAD point on a selected surface is often a requirement when defining or re-defining a measurement routine.

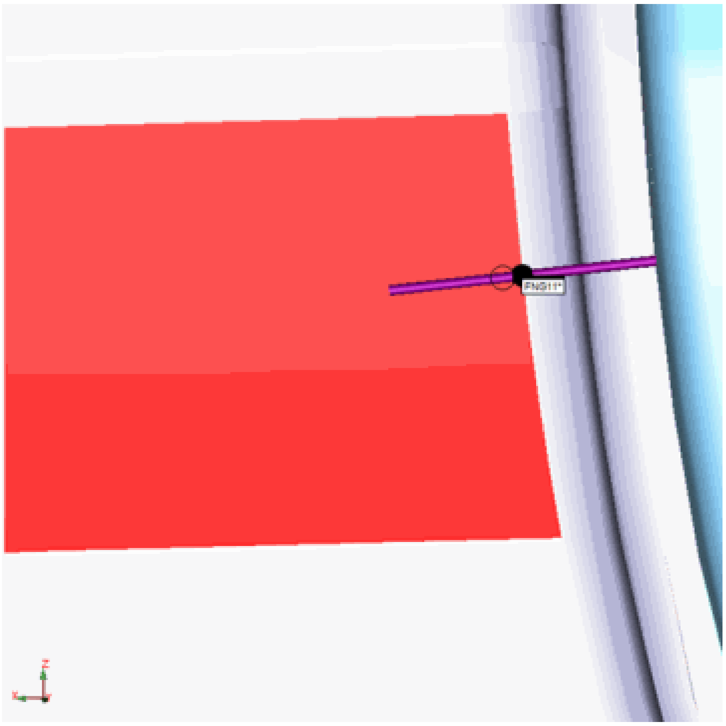
The first point clicked on in the Graphic Display window, other than the Master Side Point and the Edge Vector, is now displayed as a black circle centered in the picked point and the selected surface is highlighted.

Sometimes it happens that the Master Side Point found is in a wrong surface boundary location, and the point is required to be clicked on again. The following describes two ways this can be done:

1. If the desired Master Side Point is on the edge of the highlighted surface, then it is sufficient just to re-click a point on the surface very close to the edge.
2. If the desired Master Side Point doesn't lie on the highlighted surface, clicking the drawn circle area causes the interface to be reset. PC-DMIS is then ready for you to re-take the first point. To help re-define the new surface selection, the previous surface is left highlighted. See pictures below.







Example of Flush and Gap CAD Selection Capability

Without CAD Data

1. Move the machine to the gap location using the **Laser View** tab of the Graphic Display window.
2. Click the **Read Point from Position** button.
3. Manually type all of the theoretical XYZ and IJK values. These include the Flush and Gap **Point**, the **View Vector**, **Gap Dir** (gap direction), **Master Pnt** (master point), **Gauge Pnt** (gauge point), **Master Vec** (master vector), and **Gauge Vec** (gauge vector).
4. Note that when you change some parameters, and you don't have any CAD data, PC-DMIS adjusts some parameter values automatically. For information, see "Automatically Adjusted Flush and Gap Values".
5. Set the **Indent** and the **Spacer** values so that they only include points on the flat surfaces and not points on the curved portion.
6. Set other parameters as needed. For information, see "Flush and Gap Specific Parameters".
7. Enter the necessary information in the **Probe Toolbox**. tabs. Cycle through the **Laser Scan**, **Laser Filtering**, and **Laser Clipping**, properties tabs to enter the information.
8. If desired, click the **Test** button to test the feature.

WARNING: When you click Test, the machine moves. To avoid injury, stay clear of the laser and the machine. To avoid hardware damage, run the machine at a slower speed.

9. Click the **Create** button and then **Close**.

Flush and Gap Specific Parameters

For a visual example of these parameters, consult the diagrams below.

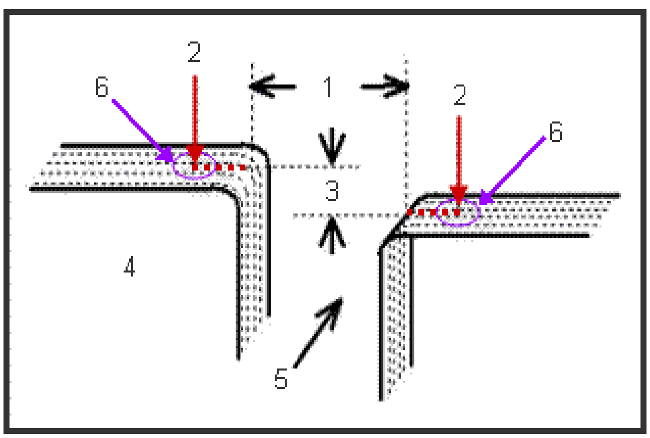
Flush - This box determines the height difference between two mating sheet metal parts. Whether or not the flush value is positive or negative depends on whether it's higher or lower than the "Master" side.

Gap - This box determines the distance (in the same plane) between two mating sheet metal parts.

Indent - The indent specifies the distance from the gap's edge at which PC-DMIS measures the flush.

Spacer - This is a circle at the indent point used to calculate the surface normals used in the calculation.

Gap Dir (Vector) - These boxes in the **Feature Properties** area define the direction of the gap.



Flush and Gap Diagram

Key:

1 - Gap

2 - Indent

3 - Flush (negative flush is shown at left)

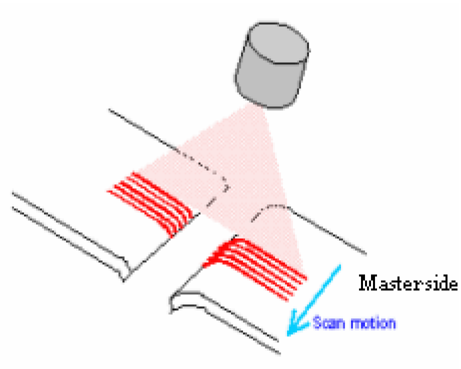
4 - Master side

5 - Cut Vector

6 - Spacer

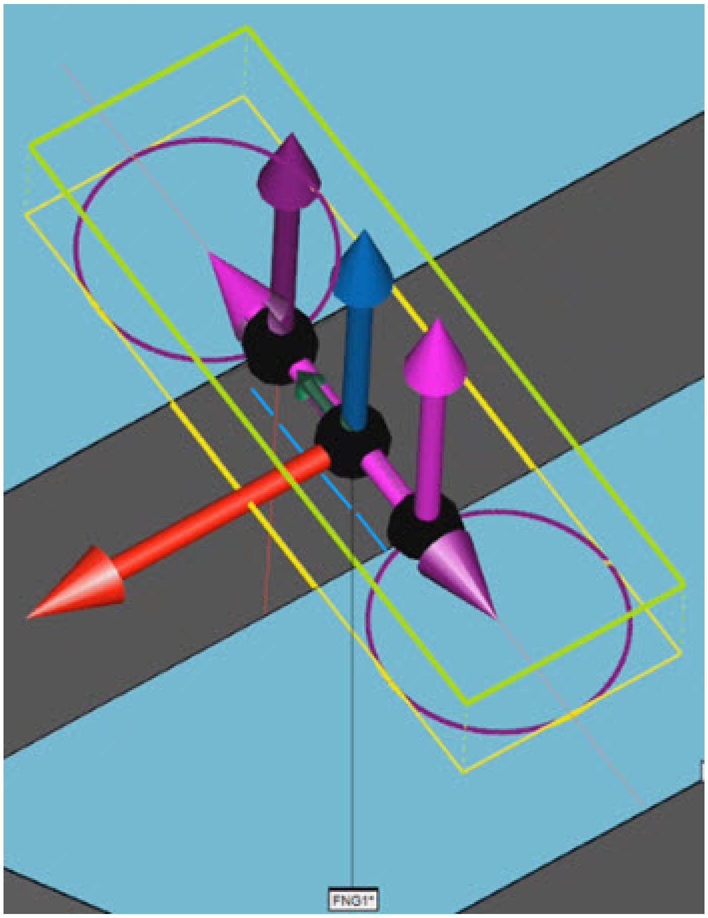
NOTE: The "Master" side is always to the left of the scan/gap direction.

The direction of the scan is controlled by the specified Cut Vector and not the direction of the laser stripe.



Direction of scan

NOTE: The "Master" side is always to the left of the Cut Vector.



Sample Flush and Gap in the Graphic Display window showing the Indent (red lines), Spacer (purple circles), Depth (blue line), Horizontal clipping region (yellow lines), Vertical clipping region (green), the View vector (blue arrow) and the Cut vector (red arrow).

Flush and Gap Command Mode Text

The Flush and Gap command inside the Edit window's Command Mode looks like this:

```
FNG2 =FEAT/LASER/FLUSH AND GAP/DEFAULT,CARTESIAN
    THEO/<124.012,13.241,0>,<0,0,1>,<1,0,0>,0,7.985
    ACTL/<124.012,13.241,0>,<0,0,1>,<1,0,0>,0,7.985
    TARG/<124.012,13.241,0>,<0,0,1>
    MASTER SIDE POINT
    THEO/<128,13.241,0>,<0,0,1>
    ACTL/<0,0,0>,<0,0,0>
```

```
GAUGE SIDE POINT
THEO/<120,13.241,0>,<0,0,1>
ACTL/<0,0,0>,<0,0,0>
CUT PLANE VECTOR<0,1,0>,<0,1,0>
Depth=1
INDENT=3
SPACER=1.5
SHOW FEATURE PARAMETERS=NO
SHOW_LASER_PARAMETERS=YES
    POINT CLOUD ID=DISABLED
    ZOOM=2A,GAIN=NORMAL,OVERLAP=1
    OVERSCAN=5
    REDUCTION FILTER=OFF
    FILTER LINES=Disabled
    CLIPPING TOP=100,BOTTOM=0,LEFT=0,RIGHT=100
    SOUND=ON
    HORIZONTAL CLIPPING=2,VERTICAL CLIPPING=5
```

Flush and Gap Graphical Analysis

The Flush and Gap analysis is comprised of these three regions. Consult the diagram at the bottom of this topic:

1. **Gap region** - In the Gap region, the points that are analyzed are in a box centered on the gap point and oriented along the Gap vector. The box's height is 60% of Gap length value. The width is 130% of Gap length value.
2. **Master Flush region** - In the Master Flush region, the points are analyzed in an area that begins at the Master Side point in a direction opposite from the Master Edge Vector. It has a length that is 60% of the Gap length value.
3. **Gauge Flush region** - In the Gauge Flush region, the points are analyzed in an area that begins at the Gauge Side point in the direction opposite from the Gauge Edge Vector. It has a length that is 60% of the Gap length value.

The Flush and Gap analysis is performed using these measured items.

- Gap Point and Vector
- Master Side Point
- Master Side Surface and Edge Vectors
- Gauge Side Point

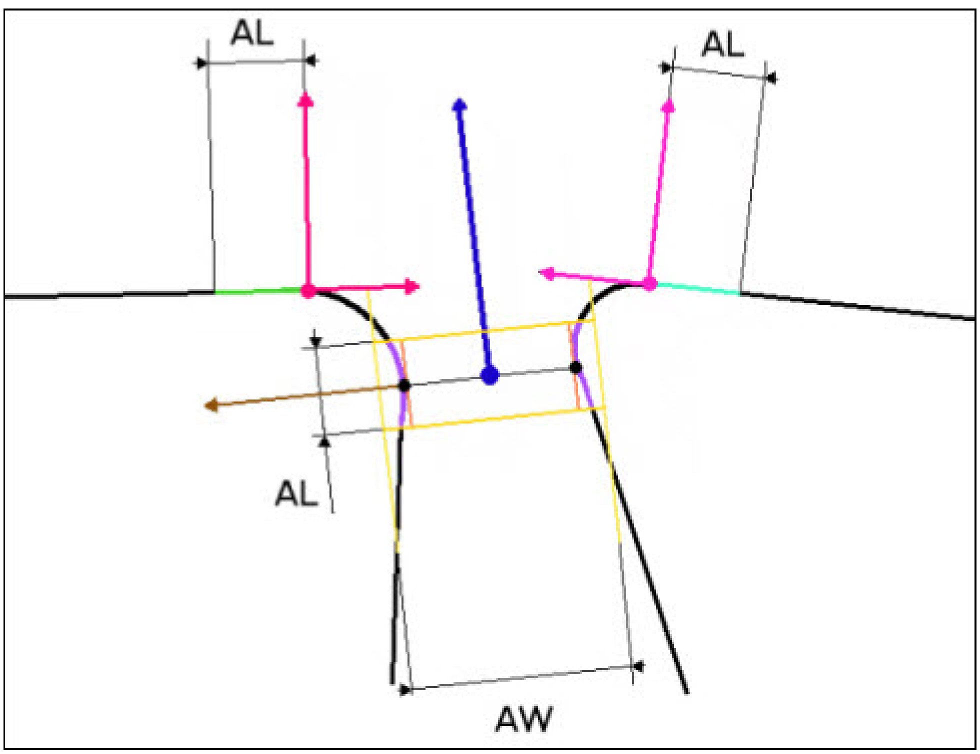
- Gauge Side Surface and Edge Vectors

PC-DMIS computes the Flush and Gap measured point's distance from these four measured reference planes:

- The first two planes are the Gap analysis reference planes defined from the two measured Minimum Distance Points (where the Gap distance is calculated) and the measured Gap Vector.
- The third plane is the measured Master Side analysis reference plane. It is defined by the measured Master Side Point and the measured Master Side Surface Vector.
- The fourth plane is the measured Gauge Side analysis reference plane. It is defined by the measured Gauge Side Point and the measured Gauge Side Surface Vector.

To reduce the analysis time, PC-DMIS only uses the points closest to the cut plane (less than 0.5 mm or 0.19685 inches).

Graphical Analysis Diagram:



Key:

AL - Analysis Length. It is 60% of the Gap length value.

AW - Analysis Width. It is 130% of the Gap length value.

- Minimum Distance Points

- Gap Vector

- Gap Point and View Vector

- Gauge Side Point and Vectors

- Master Side Point and Vectors

- Master Side Flush analysis region. Reference Plane.

- Gauge Side Flush analysis region. Reference Plane.

- Gap analysis region

- Gap analysis reference plane

Automatically Adjusted Flush and Gap Values

Note that when you change some Flush and Gap parameters, and you don't have any CAD data, PC-DMIS will adjust some parameters' values automatically. This topic details what gets changed and how the software computes those automatic values.

HINT:

Key: Use these abbreviations when viewing the equations below:

CPV = Cut Plane Vector
VV = View Vector
x = Cross Product
GV = Gap Vector
GD = Gap Distance
GP = Gap Point
GPV = Gap Point Vector

When Typing in a Gap Point Value or Modifying it by Read Position...

- The current probe vector is used as the View Vector.
- The current stripe vector is used as the Gap Vector.
- The new cut plane is located in the Gap point, and the new Cut Plane Vector is computed: $CPV = VV.x(GV)$
- Master Side Point and Gauge Side Point are ESTIMATED at $\frac{GD}{2}$ from the new Gap Point along the Gap Vector.

If the Flush distance is positive, the Master Side Point is translated along the View Vector of the flush value.

If the Flush distance is negative, the Gauge Side Point is translated along the View Vector of the flush value.

- The Master Side Surface Vector and the Gauge Side Surface Vector are set with the View Vector.

When Typing in a View Vector Value...

- The new cut plane is located in the Gap point, and the new Cut Plane Vector is computed: $CPV = VV.x(GV)$
- The Gap Vector is computed to be orthogonal to the new View Vector: $GV = CPV.x(VV)$
- The Master Side Surface Vector and the Gauge Side Surface Vector are projected onto the new cut plane.
- The Master Side Point and Gauge Side Point are projected onto the new Cut Plane.

When Typing in a Gap Vector Value...

- The new cut plane is located in the Gap point and the new Cut Plane Vector is computed: $CPV = VV.x(GV)$
- The View Vector is computed to be orthogonal to the new Gap Vector: $VV = GV.x(CPV)$
- The Master Side Surface Vector and the Gauge Side Surface Vector are projected onto the new cut plane.
- Master Side Point and Gauge Side Point are projected onto the new Cut Plane.

When Typing in a Master Side Point Value or Modifying it by Read Position...

- The new cut plane is computed orthogonal to the View Vector and the Master Side Point minus the Gap Point: $CPV = VV.x(MSP - GP)$
- The Gap Vector is computed orthogonal to the new View Vector. $GV = CPV.x(VV)$
- The Master Side Surface Vector, the Gauge Side Surface Vector and Gauge Side Point are translated onto the new cut plane.

When Typing in a Gauge Side Point Value or Modifying it by Read Position...

- The new cut plane is computed centered on the new Master Side Point and orthogonal to the View Vector and the Master Side Point minus the Gauge Side Point: $CPV = VV.x(MSP - GSP)$
- The Gap Vector is computed orthogonal to the new View Vector: $GV = CPV.x(VV)$
- The Master Side Surface Vector, the Gauge Side Surface Vector, and Gap Point are translated onto the new cut plane.

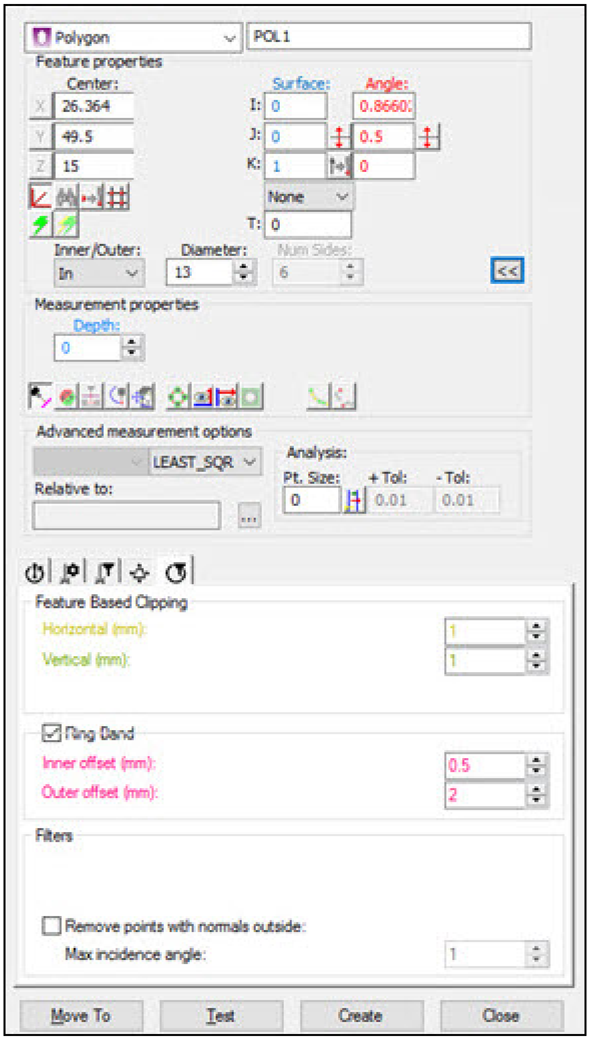
When Typing in a Flush Distance Value...

- The Master Side Point and/or the Gauge Side Point are translated according the new flush value along the Master or Gauge Side Surface Vector.

When Typing in the Distance Value...

- The Master Side Point and/or the Gauge Side Point are translated according the new gap value along the Gap Vector.

Laser Polygon



Polygon Auto Feature

NOTE: You can only use this dialog box to measure a hexagon feature (a polygon with 6 sides).

To measure a hexagon feature with a laser sensor:

1. Access the **Auto Features** dialog box, and select **Polygon**.
2. Do one of the following:

    - Perform clicks on the CAD to give the polygon a location and vector. Manually enter any remaining information.

- Move the machine to the sphere location using the **Laser View** tab of the **Graphic Display window**. Click the **Read Point from Position** button, then manually enter the any remaining information such as **Diameter**, as needed.
- Manually enter all of the theoretical X, Y, Z, I, J, K, Diameter and so on.

3. Enter the necessary information in the **Probe Toolbox** tabs. Cycle through the **Laser Scan**, **Laser Filtering**, and **Laser Clipping** properties tabs to enter the information.
4. If desired, click the **Test** button to test the feature.

WARNING: When you click Test, the machine moves. To avoid injury, stay clear of the laser and the machine. To avoid hardware damage, run the machine at a slower speed.

5. Click **Create** and then **Close**.

Polygon Specific Parameters

Num Sides - This parameter defines the number of sides used on the polygon. For Laser devices the number of sides for the Auto Feature Polygon is fixed at 6.

Diameter - The value in this box defines the polygon's diameter.

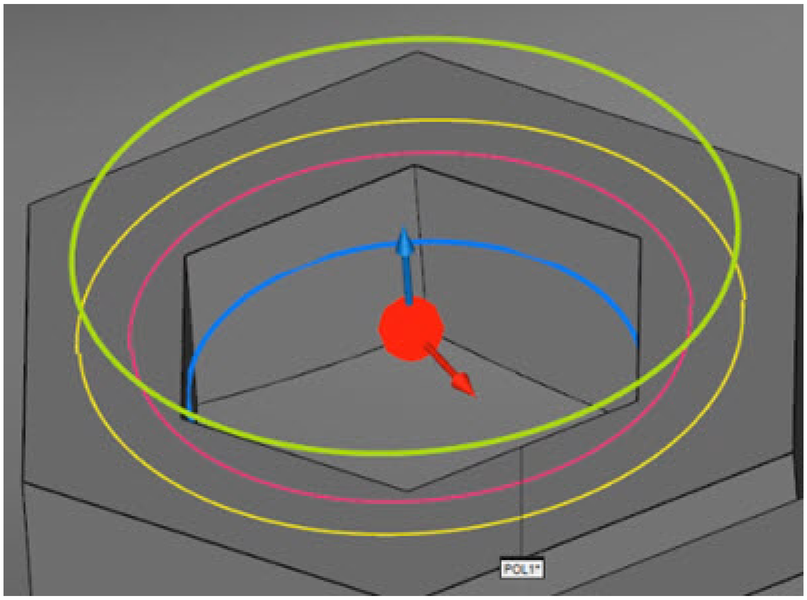
Depth - This parameter controls what data PC-DMIS uses to calculate the feature characteristics. You can use the depth value to eliminate data on a chamfer or some other transitional portion of the feature that you don't want in the feature calculation. Specifying a positive value tells PC-DMIS where along the feature PC-DMIS looks to calculate the feature characteristics. A depth of 0 causes this feature to be calculated at the surface plane height, using data found at the lowest possible depth from the surface plane. A depth of any other value causes it to be calculated at that depth. Due to hardware limitations, for this feature type if you do use a depth value higher than 0, you must use a minimum of 0.3 mm (.01181 inches).

IMPORTANT: The depth defaults to zero. This is the default value for a plane feature with no extruded edges. You should only change this to a different value if there are specific requirements from your part's drawing. Otherwise, PC-DMIS unsuccessfully attempts to locate points at the specified depth, resulting in a feature calculation error from the feature extraction module.

For example, a depth of 3 indicates that you want to use all data 3 mm (or inches, depending on the units of the measurement routine) and above for the calculation. If you specify 0, this indicates that you want to use all available data for the calculation.

For thin walled features, a 0 value may be valid; but for parts with any depth to them, you probably need to specify a depth to get accurate results.

NOTE: Even if you specify a depth greater than zero, the measured results are always projected into the plane where the feature resides.



Sample Polygon in the Graphic Display window showing:

- *the Ring Band (pink circles)*
- *the Horizontal overscan (yellow circle)*
- *the Vertical overscan (green circles)*
- *the Depth (blue)*

Polygon Command Mode Text

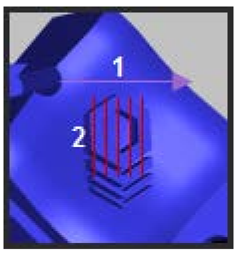
The Polygon command inside the Edit window's Command Mode looks like this:

```
POL1 =FEAT/LASER/POLYGON,CARTESIAN
    THEO/<1.0379,1.9488,0.5906>,<0,0,1>,<0.8660254,-
    0.5,0>,0.5118
    ACTL/<1.0379,1.9488,0.5906>,<0,0,1>,<0.8660254,-
    0.5,0>,0.5118
    TARG/<1.0379,1.9488,0.5906>,<0,0,1><0.8660254,-0.5,0>
    NUMSIDES=6
    DEPTH=0
    SHOW FEATURE PARAMETERS=NO
```

```
SHOW_LASER_PARAMETERS=YES
    POINT CLOUD ID=DISABLED
    SENSOR FREQUENCY=30,OVERLAP=0.0394
    OVERSCAN=0.0787,EXPOSURE=35
    FILTER=NONE
    PIXEL LOCATOR=GRAY SUM,Min=30,Max=300
    CLIPPING TOP=100,BOTTOM=0,LEFT=0,RIGHT=100
    RINGBAND=OFF
```

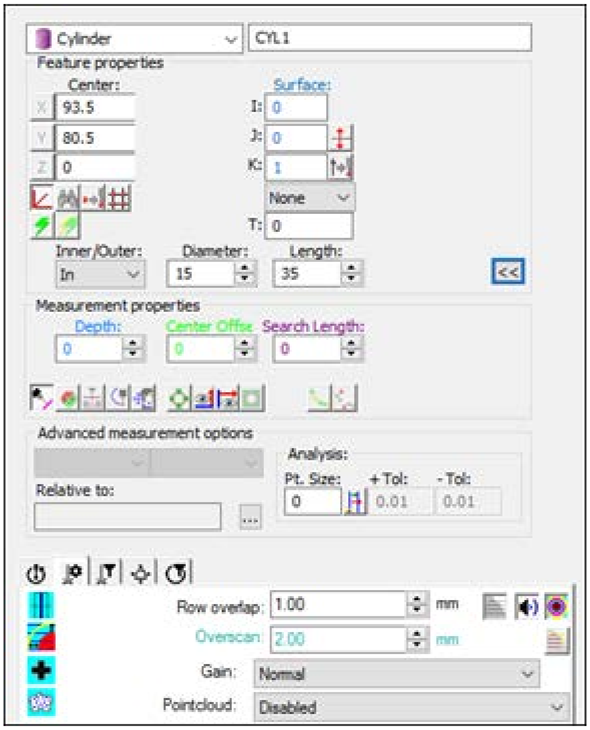
Auto Polygon Paths

PC-DMIS uses the **Angle** IJK vector to determine the direction of the scan.



The feature's scan lines or laser stripes (shown in 2) are perpendicular to the feature's angle vector (shown in 1)

Laser Cylinder



Cylinder Auto Feature

To measure a cylinder with a laser sensor:

1. Access the **Auto Features** dialog box, and select **Cylinder**.
2. From the **Inner/Outer** box, choose **In** or **Out**.
3. Do one of the following:

   - Perform clicks on the CAD to give the cylinder location and vector. Then manually enter any remaining information.
   - From the Graphic Display window, use the **Laser View** tab to move the machine to the cylinder location. Next, from the **Feature Properties** area, click **Read Point from Machine** . Then manually enter the any remaining information like the Inner/Outer value, diameter, length, and so on.
   - Manually enter all of the theoretical information for X, Y, Z, I, J, K, Inner/Outer value, diameter, length, depth, and so on.

4. Enter the necessary information in the **Probe Toolbox** tabs. Cycle through the **Laser Scan**, **Laser Filtering**, and **Laser Clipping**, properties tabs to enter the information.
5. If desired, click the **Test** button to test the feature.

WARNING: When you click Test, the machine moves. To avoid injury, stay clear of the laser and the machine. To avoid hardware damage, run the machine at a slower speed.

6. Click the **Create** button and then **Close**.

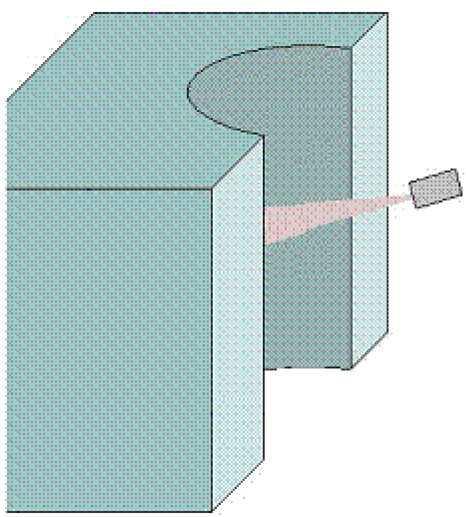
NOTE: The location and direction vector of the feature defines the center axis of the cylinder.

Cylinder Specific Parameters

Diameter - The value in this box defines the cylinder's diameter.

Length - The value in this box provides the length (height) of the cylinder's axis. The length parameter is only valid as a nominal. The software does not actually measure the length.

Inner/Outer - This parameter defines whether or not the cylinder is an inner cylinder (hole) or an outer cylinder (including a stud).



NOTE: Unlike other Laser Auto Features, for the **Overscan** value on the **Laser Scan Properties** tab of the **Probe Toolbox**, you should use negative values. This limits measurement in the cylindrical region along the cylinder axis.

Depth - This parameter controls the location of the laser focal point in relation to the cylinder's outside diameter (outer cylinders) or the cylinder's center axis (inner cylinders). This allows you to control how the laser stripes fall on the cylinder's surface because you can specify how far or close the laser is to the cylinder's surface. A depth of 0 for an internal feature means that the laser sensor center is on the cylinder center axis. For an external feature, it is on the surface of the outer cylinder.

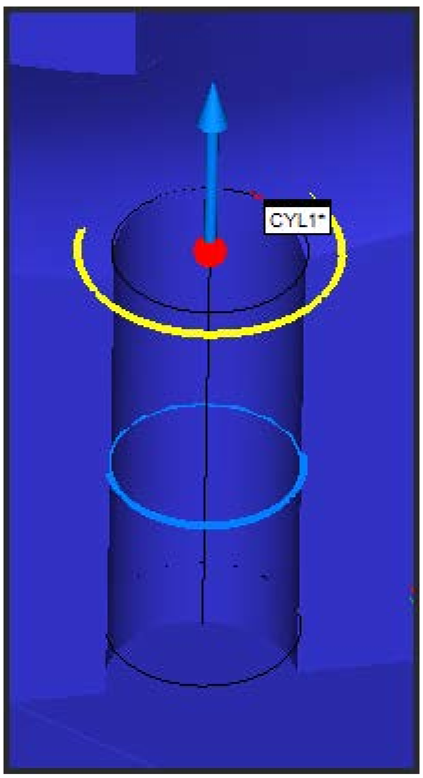
- A negative depth value moves the laser sensor's center away from the cylinder's surface.
- A positive depth value moves the laser sensor's center closer to the cylinder's surface.

Center Offset - This value identifies the center of the cylinder portion of the stud.

Search Length - This value identifies the length of the cylinder portion.

IMPORTANT: The depth defaults to zero for a plane feature with no extruded edges. You should only change this to a different value if there are specific requirements from your part's drawing. Otherwise, PC-DMIS unsuccessfully attempts to locate points at the specified depth. This results in a feature calculation error from the feature extraction module.

Sample Inner Cylinder

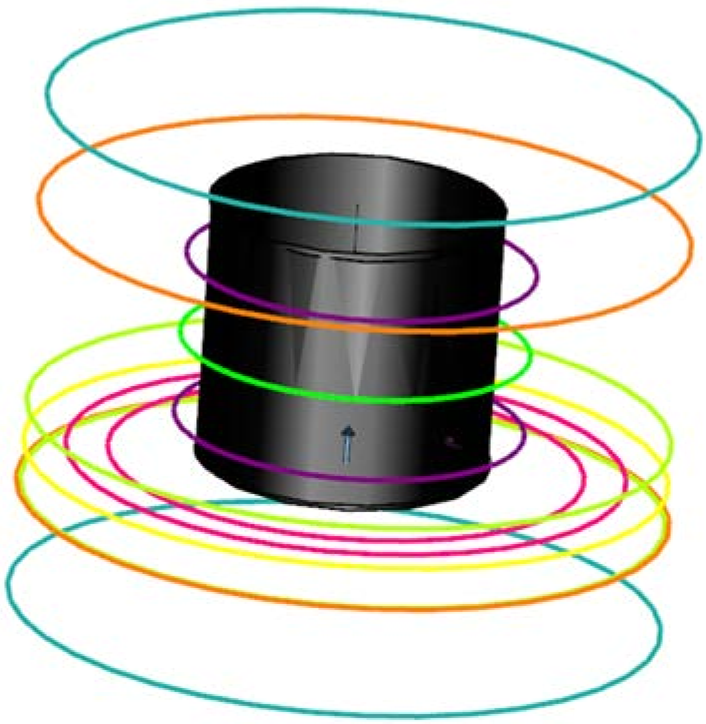


Sample inner Cylinder that shows:

- *The **Depth** (blue circle)*
- *The **Length** (bottom black circle)*

- *The **Center Point** (yellow circle)*

Sample Outer Cylinder



Sample stud Cylinder that shows:

- *The **Search Length** (purple circles)*
- *The **Center Offset** (lime green circle)*
- *The **Point Segregation** (orange circles)*
- *The **Center Point** (yellow circle)*
- *The **Clipping Plane** (light green circles)*
- The **Overscan** (sea green circles)
- *The **Ring Band** (pink circles)*

Cylinder Command Mode Text

Sample Cylinder

```
CYL1 =FEAT/LASER/CYLINDER/DEFAULT,CARTESIAN,OUT
     THEO/<3.1425,2.7539,0>,<0,0,1>,0.25,0.25
     ACTL/<3.1425,2.7539,0>,<0,0,1>,0.25,0.25
     TARG/<3.1425,2.7539,0>,<0,0,1>
     DEPTH=0
```

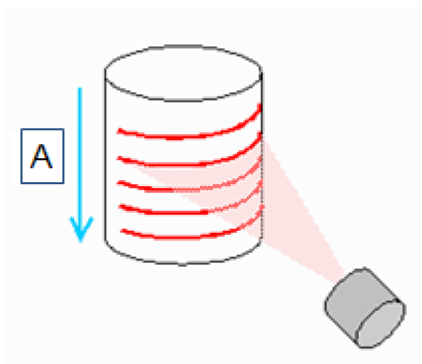
```
    CENTER OFFSET=0
    SEARCH LENGTH=0
    SHOW FEATURE PARAMETERS=NO
    SHOW_LASER_PARAMETERS=YES
        POINT CLOUD ID=COP1
        HORIZONTAL CLIPPING=0.0787,VERTICAL CLIPPING=0.0787
        RINGBAND=ON,INNER OFFSET=0.5,OUTER OFFSET=2
```

Auto Cylinder Paths

Cylinder Measurements

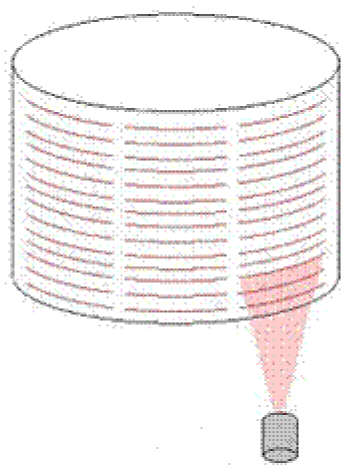
Adjust the processing window in the Laser View to include as much of the cylindrical surface as possible. Laser plane must be roughly normal to the cylinder axis (< 30 degree deviation). Depending on the diameter of the cylinder, PC-DMIS takes one of these paths when performing the measurement:

Path 1: Single Scan



Cylinders with a diameter less than the usable portion of the stripe. A is the scan motion.

Path 2: Multiple Scans

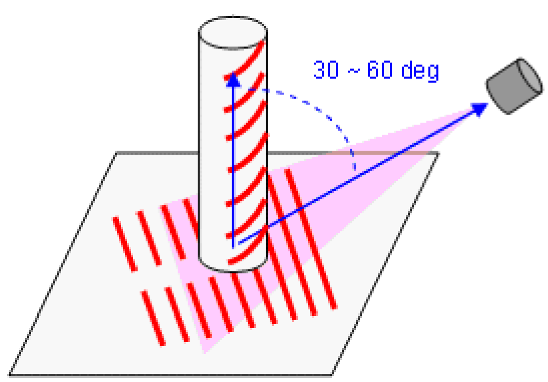


Cylinders with a diameter larger than the usable portion of the stripe

Stud Measurements

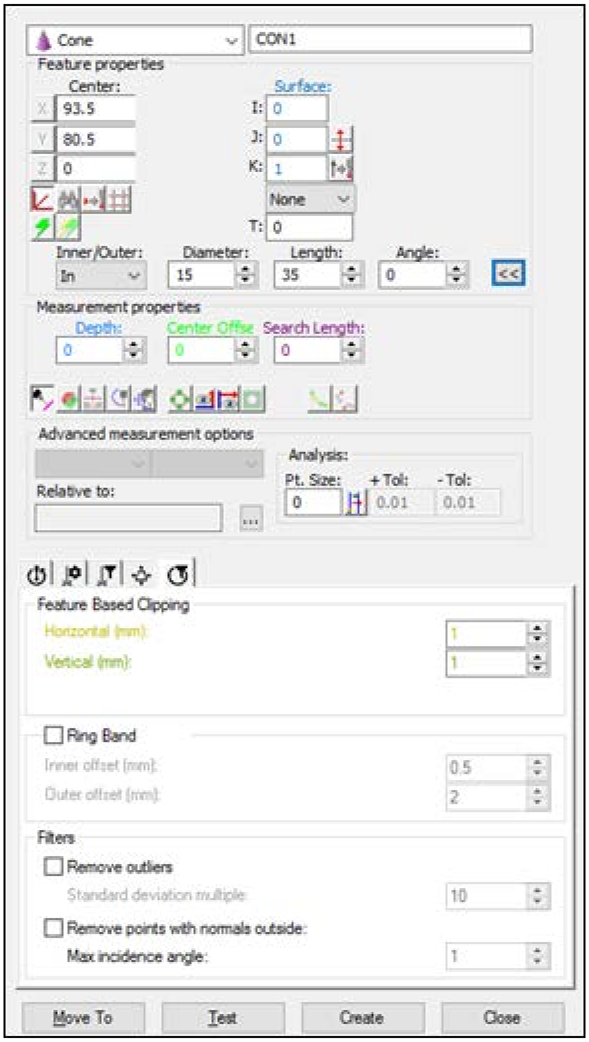
Single Scan

Adjust the processing window in the Laser View to include as much of the cylindrical surface as possible. The laser plane must be roughly 30~60 degrees to the cylinder axis. The scan must capture the region on the base plane of the stud where the cylinder is mounted.



Single pass laser scan on stud cylinder

Laser Cone



Cone auto feature

To measure a cone with a laser sensor:

1. Access the **Auto Feature** dialog box, and select **Cone**.
2. From the **Inner/Outer** box, select **In** or **Out**.
3. Do one of the following:

    - Click on the CAD to give the cone location and vector, and then manually enter any remaining information.
    - From the Graphic Display window, use the **Laser View** tab to move the machine to the cone location. Next, from the **Feature Properties** area, click the **Read Point from Position** button. Then manually enter the any

remaining information, such as the inner/outer value, diameter, length, and so on.
- Manually enter the theoretical information for X, Y, Z, I, J, K, inner/outer value, diameter, length, depth, and so on.

4. Enter the necessary information in the **Probe Toolbox** tabs. Cycle through the **Laser Scan**, **Laser Filtering**, and **Laser Clipping** properties on the tabs to enter the information.
5. If desired, click the **Test** button to test the feature.

WARNING: When you click Test, the machine moves. To avoid injury, stay clear of the laser and the machine. To avoid hardware damage, run the machine at a slower speed.

6. Click **Create** and then **Close**.

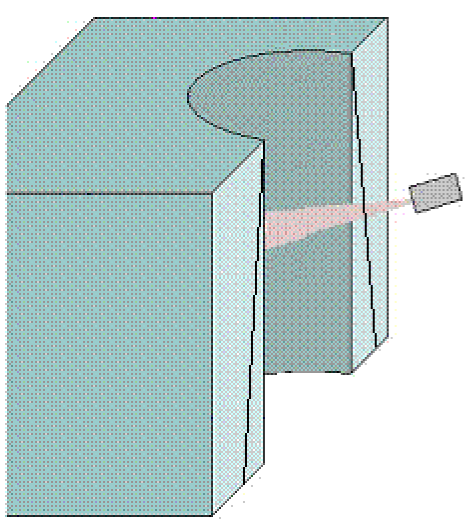
NOTE: The location and direction vector of the feature define the center axis of the cone.

Cone Specific Parameters

Diameter: The value in this box defines the cone's diameter.

Length: The value in this box provides the length (height) of the cone's axis. The length parameter is only valid as nominal. No actual length measurement will be performed.

Inner/Outer: This parameter defines whether or not the cone is an inner cone (hole) or an outer cone (stud).



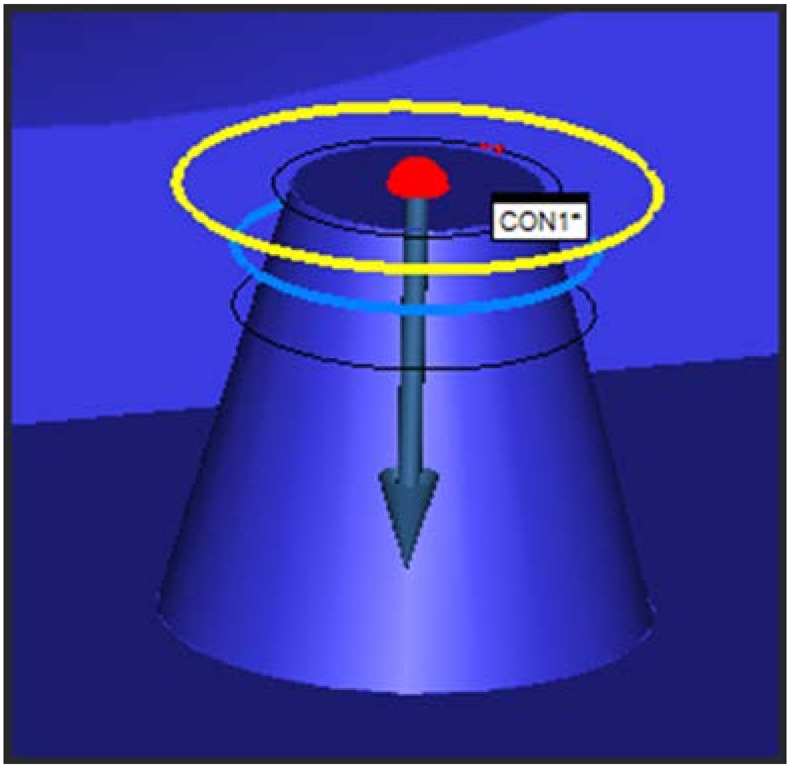
NOTE: The **Overscan** value on the **Laser Scan Properties** tab of the **Probe Toolbox** should use negative values unlike other Laser Auto Features. This limits measurement in the conical region along the cone axis.

Depth - This parameter controls the location of the laser focal point in relation to the cone's outside diameter (outer cones) or the cone's center axis (inner cones). This allows you to control how the laser stripes fall on the cone's surface by specifying how far or close the laser is to the cone's surface. A depth of 0 (zero) causes this feature to be calculated at the surface plane height, using data found at the lowest possible depth from the surface plane. A depth of any other value causes it to be calculated at that depth.

Center Offset - This value identifies the center of the cone portion of the stud.

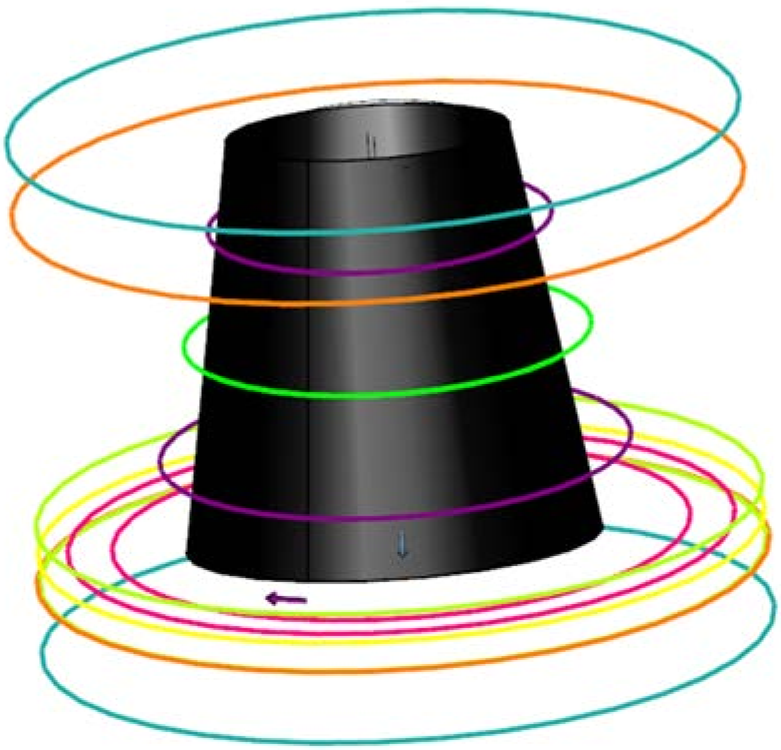
Search Length - This value identifies the length of the cone portion.

IMPORTANT: The depth defaults to 0 (zero). This is the default value for a plane feature with no extruded edges. You should only change this to a different value if there are specific requirements from your part's drawing. Otherwise, PC-DMIS unsuccessfully attempts to locate points at the specified depth, resulting in a feature calculation error from the feature extraction module.



Sample external Cone in the Graphic Display window showing:

- *The **Diameter** (top black circle)*
- *The **Length** (bottom black circle)*
- *The **Depth** (blue circle)*
- *The **Center Point** (yellow circle)*



Sample external Stud Cone in the Graphic Display window showing:

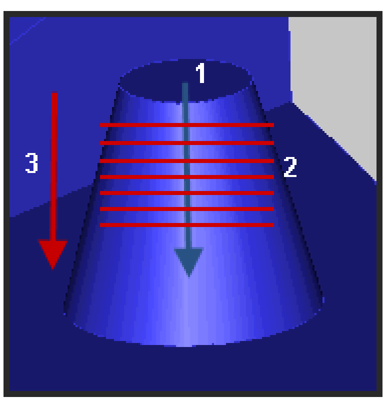
- *The **Search Length** (purple circles)*
- *The **Center Offset** (lime green circle)*
- *The **Point Segregation** (orange circles)*
- *The **Center Point** (yellow circle)*
- *The **Clipping Plane** (light green circle)*
- *The **Overscan** (sea green circles)*
- *The **Ring Band** (pink circles)*

Cone Command Mode Text

```
CON1 =FEAT/LASER/CONE/DEFAULT,CARTESIAN,OUT
    THEO/<3.1425,2.7539,0>,<0,0,1>,0.5,20,12.7
    ACTL/<3.1425,2.7539,0>,<0,0,1>,0.5,20,12.7
    TARG/<3.1425,2.7539,0>,<0,0,1>
    DEPTH=0
    CENTER OFFSET=3
    SEARCH LENGTH=2
    SHOW FEATURE PARAMETERS=YES
        SURFACE=THEO_THICKNESS,0
        RMEAS=NONE,NONE,NONE
        AUTO WRIST=YES
        GRAPHICAL ANALYSIS=NO
    SHOW_LASER_PARAMETERS=YES
        POINT CLOUD ID=COP1
            SOUND=OFF
        HORIZONTAL CLIPPING=0.0787,VERTICAL CLIPPING=0.0787
    RINGBAND=ON,INNER OFFSET=0.5,OUTER OFFSET=2
    OUTLIER_REMOVAL=ON,1
```

Auto Cone Paths

The laser sensor scans along the length of the cone. It moves in the direction of the cone's vector. The laser must be nearly perpendicular to that vector.

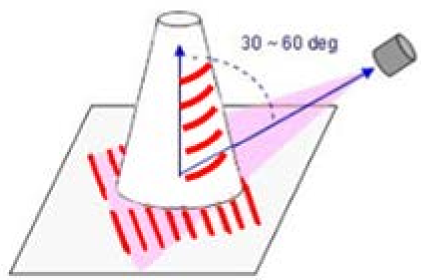


1 - The feature's vector. 2 - The feature's scan lines or laser stripes are perpendicular to the feature's vector. 3 - Scan direction follows the feature's vector

Stud Measurements

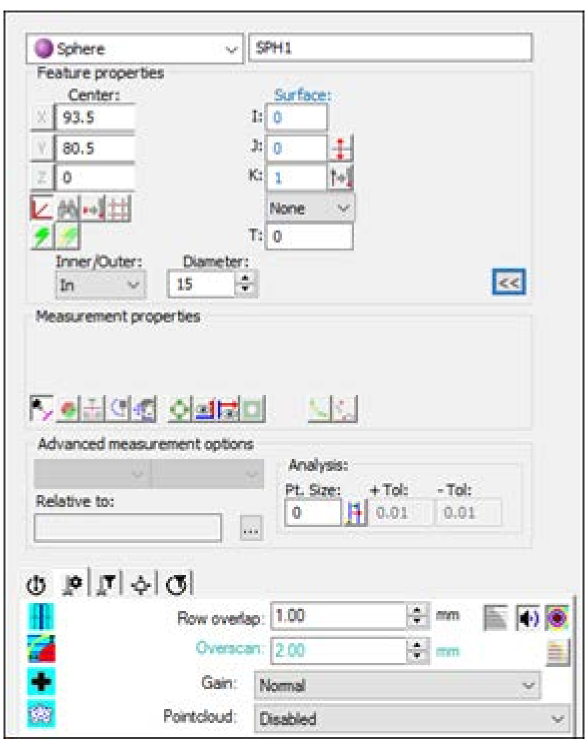
Single Scan

Adjust the processing window in the Laser View to include as much of the cone surface as possible. The laser plane must be roughly 30-60 degrees to the cone axis. The scan must capture the region on the base plane of the stud where the cone is mounted.



Single pass laser scan on stud cone

Laser Sphere



Sphere Auto Feature

To measure a sphere with a laser sensor:

1. Access the **Auto Features** dialog box, and select **Sphere**.
2. From the **Inner/Outer** box, choose **In** or **Out**.
3. Do one of the following:

    - Clicks on the CAD to give the sphere a location and vector. Then manually enter any remaining information.
    - From the Graphic Display window, use the **Laser View** tab to move the machine to the sphere. Next, from the **Feature Properties** area, click the **Read Point from Position** button. Then manually enter the any remaining information, such as the Inner/Outer value, diameter, and so on.
    - Manually enter all of the theoretical X, Y, Z, I, J, K, Inner/Outer value, diameter, and so on.

4. Enter the necessary information in the **Probe Toolbox** tabs. Cycle through the **Laser Scan**, **Laser Filtering**, and **Laser Clipping**, properties tabs to enter the information.
5. If desired, click the **Test** button to test the feature.

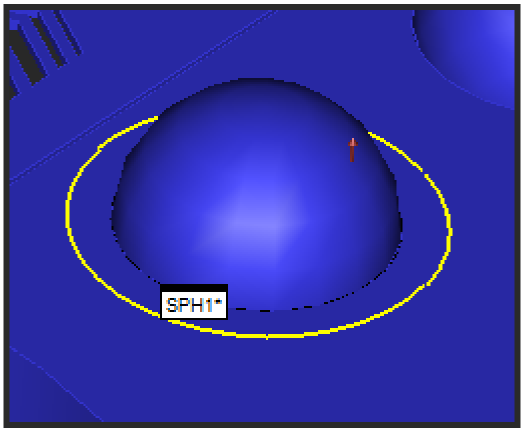
WARNING: When you click Test, the machine moves. To avoid injury, stay clear of the laser and the machine. To avoid hardware damage, run the machine at a slower speed.

6. Click **Create** and then **Close**.

Sphere Specific Parameters

Inner/Outer: This parameter defines whether or not the sphere is an inner sphere (concave) or an outer sphere (convex).

Diameter: The value in this box defines the sphere's diameter.



Sample outer Sphere in the Graphic Display window showing the Overscan (yellow circle)

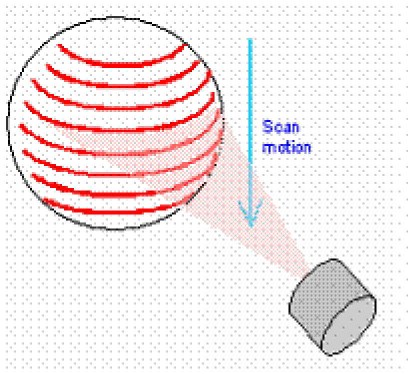
Sphere Command Mode Text

The Sphere command inside the Edit window's Command Mode looks like this:

```
SPH1 =FEAT/LASER/SPHERE,CARTESIAN,IN,LEAST_SQR
    THEO/<1.895,1.91,1>,<0,0,1>,1.895
    ACTL/<1.895,1.91,1>,<0,0,1>,1.895
    TARG/<1.895,1.91,1>,<0,0,1>
    START ANGLE 1=0,END ANG 1=0
    START ANGLE 2=0,END ANG 2=0
    SHOW FEATURE PARAMETERS=YES
        SURFACE=THEO_THICKNESS,0
        MEASURE MODE=NOMINALS
        RMEAS=NONE,NONE,NONE
        AUTO WRIST=NO
        GRAPHICAL ANALYSIS=NO
        FEATURE LOCATOR=NO,NO,""
    SHOW_LASER_PARAMETERS=YES
        POINT CLOUD ID=DISABLED
        SENSOR FREQUENCY=25,OVERSCAN=2,EXPOSURE=18
        FILTER=NONE
```

Auto Sphere Path

The direction of the path will be determined based on the stripe.



Path direction of scan

Clearing Auto Feature Scan Data

PC-DMIS's Laser Auto Features sometimes store scanned data as internal clouds of points following their creation. This occurs if the Point Cloud parameter on the Laser Scan Properties tab is set to **Disabled**.

Two menu items exist to clear out this internal data based on your needs. Located under the **Operations | Laser Autofeatures** sub menu, these menu items remove the internal data, thereby helping to reduce measurement routine size:

- **Clear All Scan Data Now** - This menu item, once selected, immediately deletes all the internal clouds of points from all the laser Auto Features in the measurement routine.
- **Clear All Scan Data after Execution** - This menu item can take a check mark. By default, this menu item is unmarked but becomes marked when you first select it. If marked then any laser auto feature that executes will delete its internal clouds of points data following execution.

NOTE: This only operates on internal clouds of points from Auto Features. It does not affect COP commands in the measurement routine.

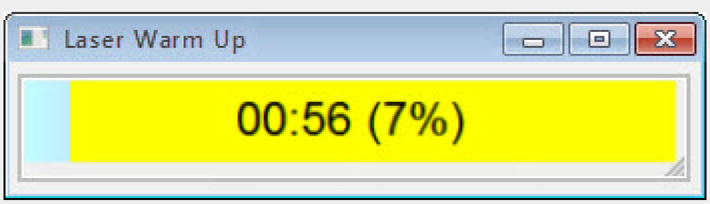
Scanning Your Part Using a Laser Sensor

When you scan the surface of the part with a laser sensor, you can define an area of measurement. The software collects a group of point data that it passes to the reference Point Cloud object in the measurement routine. When you work with point clouds and scans, note that the scans themselves DON'T contain any data. They only define the machine movement. The Point Cloud object always stores the point data.

The main topics in this section discuss the scanning options available from the **Insert | Scan** submenu when you use a laser sensor:

- Introduction to Performing Advanced Scans
- Common Functions of the Scan Dialog Box
- Performing a Linear Open Advanced Scan
- Performing a Patch Advanced Scan
- Performing a Perimeter Advanced Scan
- Performing a Freeform Advanced Scan
- Performing a Grid Advanced Scan
- Performing a Manual Laser Scan on DCC Machines
- Setting Machine Speed for Scanning
- CWS Parameter Probe Toolbox Dialog

HINT: If you use an HP-L-20.8 laser sensor, it requires a certain amount of time to reach its optimal temperature after initialization. After the sensor has been initialized and if the laser sensor is not at the optimal temperature, PC-DMIS displays the **Laser Warm Up** dialog box. This shows the remaining time to reach the appropriate temperature.



The dialog box only appears if the laser sensor needs to warm up.

Introduction to Performing Advanced Scans

Advanced scans are DCC continuous-motion scans that follow a predefined path. PC-DMIS follows the predefined path regardless of the shape of the actual part. The path can be defined in several ways that are explained later.

These advanced scans use a laser scanning probe. This enables you to automatically digitize surfaces.

To perform an advanced scan:

1. Specify the necessary parameters for the DCC scan that you selected.
2. Click the **Generate** button. PC-DMIS generates the scan.
3. Once it finishes, click the **Create** button. The PC-DMIS scanning algorithm then takes control of the measurement process.

The types of advanced scans that PC-DMIS supports include:

- Linear Open Scan
- Patch Scan
- Perimeter Scan
- Freeform Scan
- Grid Scan
- Manual Laser Scan on DCC Machines

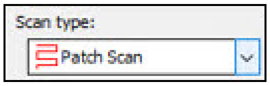
This document first covers the common functions that are available in the **Scan** dialog box (the dialog box that you use to perform these scans). It then describes how to perform the available advanced scans.

For information on setting your machine's scan speed, see "Setting the Machine Speed for Scanning".

Common Functions of the Scan Dialog Box

Many of the functions described below are common to both DCC and Manual scans. Functionality that relates specifically to one scan mode is appropriately indicated.

Scan Type



Scan type list

Use the **Scan type** list in the **Scan** dialog box to change scan types without closing the dialog box and selecting a different scan type.

ID

The **ID** box in the **Scan** dialog box displays the ID of the scan to be created.

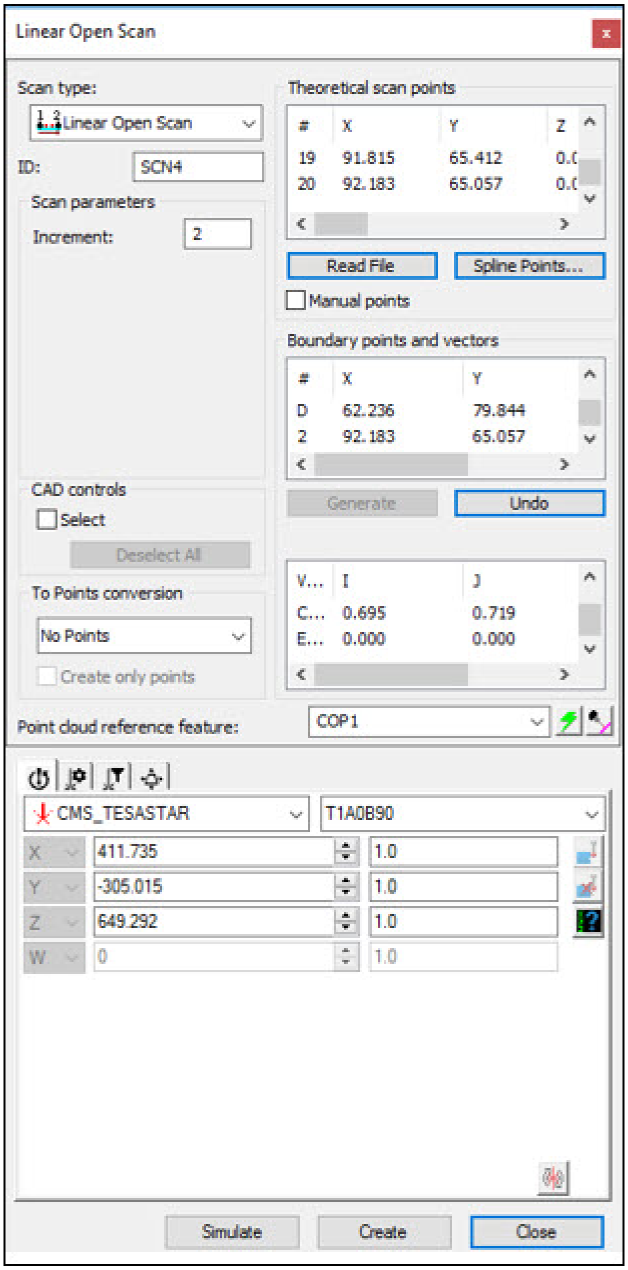
Scan Parameters

The **Scan parameters** area in the **Scan** dialog box provides different controls depending on the type of scan that is being performed. See the specific topics located under each scan type:

- Linear Open Scan Parameters
- Patch Scan Parameters
- Perimeter Scan Parameters
- Grid Scan Parameters

CAD Controls

Click the **Advanced >>** button in the **Scan** dialog box to display the full dialog box if necessary.



Scan dialog box for Linear Open Scan

Click the Graphics tab to view the CAD controls. This area enables you to specify the CAD surface elements used to define the “Theoretical Points”.

CAD controls area

In some cases, a scan might start over a certain surface and travel over many other surfaces before completion. In such cases, PC-DMIS does not know which CAD elements are to be used to generate the scan. It must therefore search through every surface in the CAD model. If the CAD model has many surfaces, it might take a long time before the scan generation is successful.

IMPORTANT: To use this functionality to select CAD surfaces, you must have the ability to import and use CAD surface data. Ensure that you select the **Draw Surfaces** button , or else when you click on the CAD model, the nearest wire gets selected instead of the selected surface.

To avoid this delay:

1. Select the **Select** check box.
2. Click on the appropriate surfaces. Once a CAD surface is selected, it is highlighted in the Graphic Display window. The status bar displays the number of surfaces that have been selected.

If you mistakenly select a surface, click on that surface a second time. This deselects the surface. Clicking the **Deselect All** button deselects all highlighted surfaces at once.

Once you are done selecting surfaces, clear the **Select** check box. The selection will be kept.

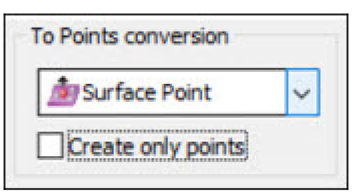
When the **Select** check box is cleared, PC-DMIS assumes any clicks on the surface to be those to create the scan path.

The following options are available:

Select check box - Enables you to select CAD surface and wire frame elements used to find the nominal.

Deselect All button - Deselects all highlighted surfaces at once that were created using the **Select** check box.

To Points Conversion



To Points conversion area

The **To Points conversion** area in the **Scan** dialog box enables you to create Point laser commands. The commands start from the points that make up the scan.

Hit Type List

The default setting is **No Points**.

For a Perimeter scan, you can select either Surface Point or Edge Point in the list. For all other types of scans, you can select only Surface Point.

The points are collected in a collapsed `GROUP` command. The name of the command includes the name of the related scan, the pointcloud associated with it, and the point ID preceded by "Edge" (if you selected Edge Point).

Surface Point Group Command Mode Text

Following is an example of a collapsed `GROUP` command collecting Surface Points:

```
COP           = COP/DATA,TOTAL SIZE=468492,REDUCED SIZE=468492,
               FINDNOMS=NO,REF,SCN1,,
SCN1           =  FEAT/SCAN,PERIMETER,NUMBER OF HITS=4,
               SHOW HITS=NO,SHOWALLPARAMS=NO,POINTCLOUDID=COP
               MEAS/SCAN
               BASICSCAN/PERIMETER,NUMBER OF HITS=4,
               SHOW HITS=NO,SHOWALLPARAMS=NO
               ENDSCAN
               ENDMEAS/
SCN1_COP_PNT_GRP1=GROUP/SHOWALLPARAMS=NO
               EXECUTION CONTROL=AS MARKED
               ENDGROUP/ID=SCN1_GRP1
```

Following is an example of a `GROUP` command collecting Edge Points:

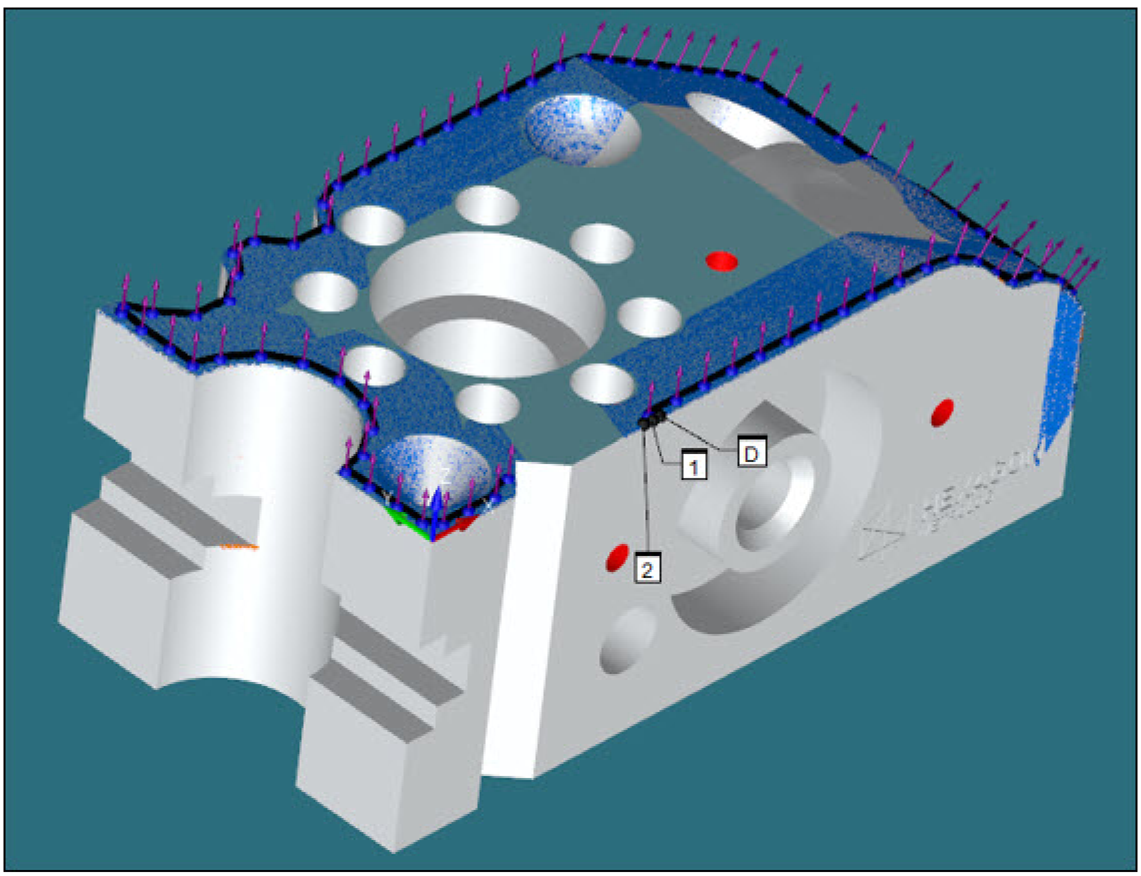
```
SCN2        =FEAT/SCAN,PERIMETER,NUMBER OF HITS=3,SHOW
HITS=NO,SHOWALLPARAMS=NO,POINTCLOUDID=COP
            MEAS/SCAN
            BASICSCAN/PERIMETER,NUMBER OF HITS=3,SHOW
HITS=NO,SHOWALLPARAMS=NO
            ENDSCAN
```

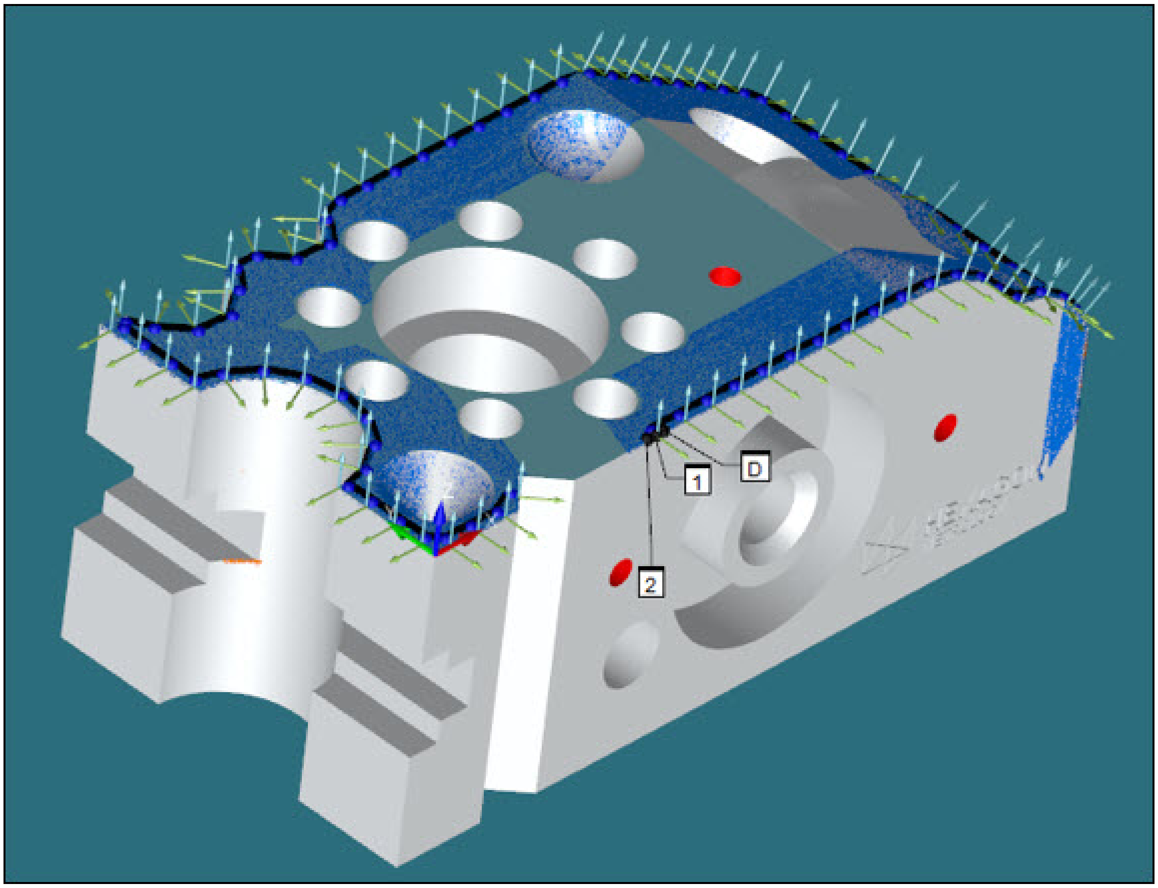
```
            ENDMEAS/
SCN2_COP_EDGEPNT_GRP2=GROUP/SHOWALLPARAMS=YES
            EXECUTION CONTROL=AS MARKED
PNT5         =FEAT/LASER/EDGE POINT/DEFAULT,CARTESIAN
            THEO/<133.992,0,0>,<0,-1,0>,<0,0,1>
            ACTL/<133.992,0,0>,<0,-1,0>,<0,0,1>
            TARG/<133.992,0,0>,<0,-1,0>,<0,0,1>
            DEPTH=0
            INDENT=1.5
            SPACER=0.5
            SHOW FEATURE PARAMETERS=NO
            SHOW_LASER_PARAMETERS=YES
              POINT CLOUD ID=COP
              SOUND=OFF
              HORIZONTAL CLIPPING=3,VERTICAL CLIPPING=3
              REMOVE POINTS WITH NORMALS OUTSIDE=ON,10
PNT6        =FEAT/LASER/EDGE POINT/DEFAULT,CARTESIAN
            THEO/<138.992,0,0>,<0,-1,0>,<0,0,1>
            ACTL/<138.992,0,0>,<0,-1,0>,<0,0,1>
            TARG/<138.992,0,0>,<0,-1,0>,<0,0,1>
            DEPTH=0
            INDENT=1.5
            SPACER=0.5
            SHOW FEATURE PARAMETERS=NO
            SHOW_LASER_PARAMETERS=YES
              POINT CLOUD ID=COP
              SOUND=OFF
              HORIZONTAL CLIPPING=3,VERTICAL CLIPPING=3
              REMOVE POINTS WITH NORMALS OUTSIDE=ON,10
PNT7        =FEAT/LASER/EDGE POINT/DEFAULT,CARTESIAN
            THEO/<143.992,0,0>,<0,-1,0>,<0,0,1>
            ACTL/<143.992,0,0>,<0,-1,0>,<0,0,1>
            TARG/<143.992,0,0>,<0,-1,0>,<0,0,1>
            DEPTH=0
            INDENT=1.5
```

```
SPACER=0.5
SHOW FEATURE PARAMETERS=NO
SHOW_LASER_PARAMETERS=YES
  POINT CLOUD ID=COP
  SOUND=OFF
  HORIZONTAL CLIPPING=3,VERTICAL CLIPPING=3
  REMOVE POINTS WITH NORMALS OUTSIDE=ON,10
ENDGROUP/ID=SCN2_COP_EDGEPNT_GRP2
```

NOTE: Surface Points and Edge Points are extracted from the COP that you specified in the scan.

Consider the following figures that show Surface Points and Edge Points extracted from a COP using the **Scan** dialog box for a Perimeter Scan:





Create only points

If you select the **Create only points** check box, PC-DMIS does not create the scan command. In this case, the `GROUP` command does not contain the name of the scan.

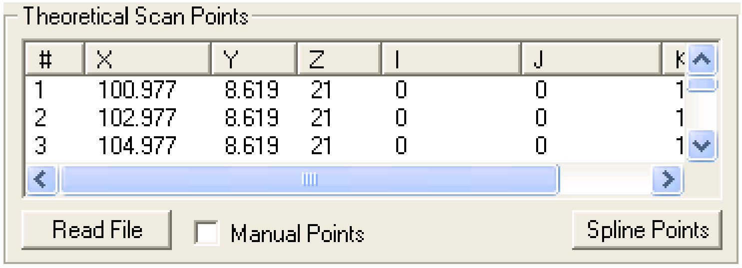
NOTE: The `SCAN` command precedes the `GROUP` command in the Edit window if you create both commands.

Theoretical Scan Points Area

You can define the Theoretical Points of a scan, through:

- Reading them from a file
- Reading machine positions
- Generating them from the defined boundary points
- Using CAD data

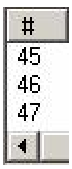
These topics are discussed in greater detail later in this section.



Theoretical Scan Points area

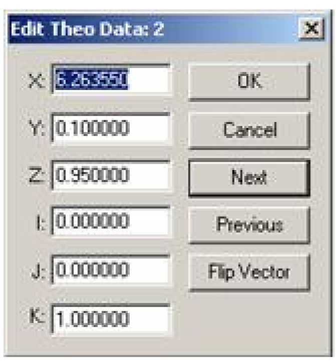
Editing Theoretical Points

To edit theoretical points double click the number of the desired point in the # column.



 # column

This displays the **Edit Theo Data** dialog box. Use this dialog box to edit the X, Y, Z, I, J, K values. The dialog box's title bar displays the ID of the point you are editing.



Edit Theo Data dialog boxes depicting Next, Previous, and Flip Vector buttons

You can cycle between the theoretical points by clicking the **Next** or **Previous** buttons.

You can also flip the vector for the selected point by clicking the **Flip Vector** button.

Deleting Theoretical Points

You can easily clear the **Theoretical Points** list of any of the scan types. Right-click inside the **Theoretical Points** list. A **Reset Theoretical Points** prompt appears. Click the prompt to clear any points from the list.

Read File

The **Read File** button tells PC-DMIS to read the Theoretical points in from a text file. The points must be in X,Y,Z,I,J,K comma delimited format. A blank space between points denotes the start of a new scan line.

Manual Points

By selecting the **Manual Points** check box, you can manually add points into the Theoretical Points list. These points can be taken by moving the probe to the desired location and Clicking the **Probe Enable** button on your jog box or by clicking points on the CAD file.

New Line

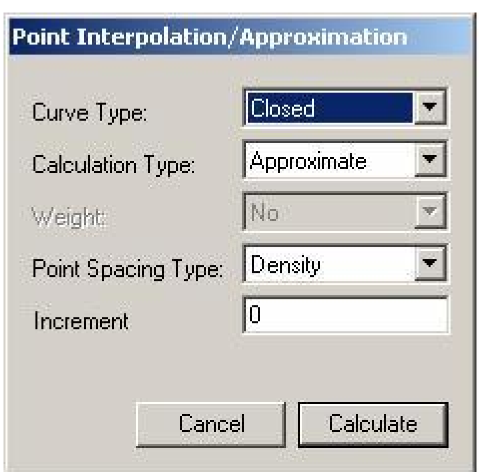
The **New Line** check box functions only for Patch Scans. By selecting the **New Line** check box, you tell PC-DMIS that manual points you take should begin a new line.

Spline Points

When you take manual points the spacing and path are usually inconsistent. With the **Spline Points** button, however, you can construct a spline curve along a path through a list of manual points and create a smooth, evenly spaced path. For a Linear Open scan PC-DMIS places all the points on the Cut Plane. For a Patch Scan it places the points for each scan line on the Cut Plane for that scan line.

NOTE: The **Spline Points** button is not available for a Perimeter scan.

Clicking the **Spline Points** button displays the **Point Interpolation / Approximation** dialog box.



Point Interpolation/Approximation

Curve Type

There are three types of curves that can be constructed with the spline routines:

Open - This option creates an open ended curve. This means the curve starts in one location and ends in another.

Closed - This option creates a close ended curve. This means the curve starts and ends in the same location.

Line - This option differs from the **Open** or **Closed** options. It doesn't use theoretical points but instead uses boundary points and creates straight lines within the boundary points, following the boundary points' direction rules.

Calculation Type

There are two calculation types you can use in spline routines.

Approximate: This option allows the path to deviate from the actual input point by a small amount in order to produce a smooth curve from which the new points are taken.

Interpolate: This option forces the curve to go through each of the input points exactly.

Weight

This list becomes available when you select the **Approximate** calculation type. When constructing the curve, it allows more weight to be given to points that are further apart. The two choices for the option are **YES** and **NO**.

Point Spacing Type

This option allows you to control the output points of the spline routine.

Density: This option lets you specify the incremental distance between each output point. PC-DMIS determines the number of output points by the length of the curve and the user supplied increment.

Number of Hits: This option lets you specify how many points they want in the output. No matter the length of the curve, PC-DMIS evenly spaces the user supplied points over the length of the curve.

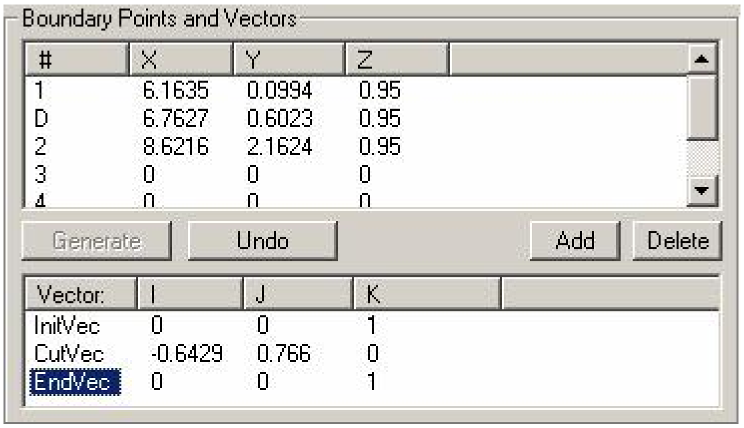
Increment

This box holds the increment value for the Point Spacing Type; either **Density** or **Number of Hits**.

Boundary Points Area

PC-DMIS lets you define the boundary of a scan. You can do this in these ways:

- Type the XYZ values for the individual boundary points directly
- Measure the points using the laser sensor
- Use the CAD data



Boundary Points and Vectors area

NOTE: Boundary Points are not available or necessary for Free-form scans

You can change the column widths of the **Boundary Point** list if you click and drag the right or left edge of a column header to the desired size. The software saves this information to your PC-DMIS Settings Editor each time it changes.

Setting Boundary Points by Typing

To set the boundary of a scan by typing:

1. Double click the desired boundary point in the '#' column to display the **Edit Scan Item** dialog box.

Edit Scan Item dialog box

2. Manually edit the X, Y or Z value.

3. Click the **OK** button to apply the changes.

Click the **Cancel** button to disregard any changes been made and close the dialog box.

Click **Next** to accept the changes and display the next boundary point for editing.

Setting Boundary Points Using the Measured Point Method

To set the boundary of the scan using measured points:

1. Place the laser sensor at the desired location.
2. On the jog box, press the **Probe Enable** button (only available on DEA and Brown and Sharpe machines).

    - This automatically updates the value of the selected boundary point in the **Boundary Points and Vectors** list. The software then selects the next boundary point (if any) in the list.
    - In the case of a PATCH scan, PC-DMIS adds an extra boundary point automatically if the selected point is the last point in the list. The PATCH scan displays the last point (this is the same as the previous point). PC-DMIS deletes this last point when you click the **OK** button.

HINT: The **Probe Enable** light on the jog box alternates between off and on every time you press the **Probe Enable** button. This is not important and has no influence on the probe itself.

Setting Boundary Points Using the CAD Data Method

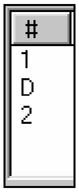
PC-DMIS lets you select the boundary points by using surface CAD data.

When using CAD surface data:

1. Make sure that you have imported solid CAD data.
2. Select that the **Draw Surfaces** icon .
3. Select a boundary point by clicking on the desired location in the Graphic Display window. PC-DMIS highlights the selected surface and automatically updates the value of the currently selected Boundary point. PC-DMIS then moves the focus to the next boundary point (if any are available). For PATCH scans, PC-DMIS automatically adds an extra boundary point if the current point is the last point in the list.

Editing Boundary Points

Boundary points can be edited by double clicking the number of the desired point in the '#' column.



 # column

This will display the **Edit Scan Item** dialog box allowing you to edit the X, Y, Z values.

Edit Scan Item dialog box

Clearing Boundary Points

You can easily clear the **Boundary Points** list of any of the scan types.

1. Right-click while the cursor is inside the **Boundary Points** list.
2. Click the **Reset Boundary Points** button that appears to reset all the boundary points to zero. The number of boundary points is set to the minimum for each scan type.

Generate

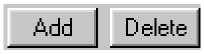
The **Generate** button is available only for DCC scans using CAD data.

After the boundary points for a scan has been defined, click the **Generate** button. PC-DMIS slices the CAD with the plane defined by the start point and cut vector and then generate the theoretical points from the curve defined by this slice. If the **Create** button is then clicked, PC-DMIS inserts a scan with nominal hit data is inserted into the measurement routine.

Undo

The **Undo** allows you to remove the hits that have been generated using the **Generate** button as outlined in the Generate topic.

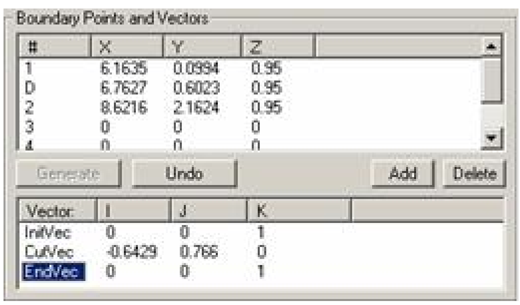
Adding and Deleting Boundary Points



Add / Delete buttons

The **Add** and **Delete** buttons allow you to add or delete boundary points to the list of boundary points. There are some restrictions regarding each type of scan. For example, a LINEAROPEN scan only takes a Start Point, a Direction point and an Ending Point. It will not allow you to add more points or delete these points. Refer to each scan for specific restrictions.

Vectors Area



Boundary Points and Vectors area

The bottom portion of the **Boundary Points and Vectors** area displays a list of vectors that PC-DMIS will use to start and stop a scan. Some of the vectors listed below may not be found in the list for a specific scan, indicating that they are not used for that scan. Please refer to each scan for more details. You can edit each of these vectors by double-clicking on the vector to edit in the vector column.

Vector column

This will display the **Edit Scan Item** dialog box:

Edit Scan Item dialog box

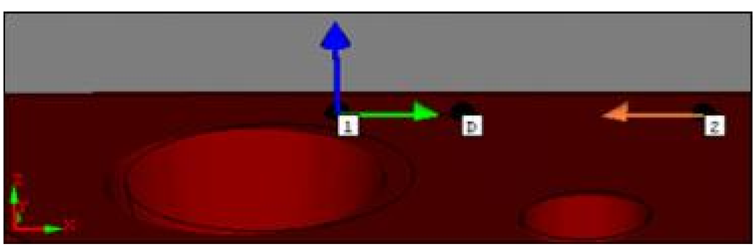
Using the different fields, you can edit the I, J, and K values.

- Clicking the **OK** button on the **Edit Scan Item** dialog box will apply any changes that have been made.
- Clicking the **Cancel** button will close the **Edit Scan Item** dialog box without applying any changes.
- Clicking the **Next** button cycles through the available vectors in the **Initial Vectors** list. Some of the initial vectors can be flipped. If so, then the **Flip** button becomes available on the **Edit Scan Item** dialog box.
- Clicking the **Flip** button allows you to reverse the direction of the selected vector.

Graphical Representation of Vectors

When setting up the start, direction, and end points of the scan, PC-DMIS allows you to see a graphical representation of the initial touch vector, the direction vector, and the vector that is normal to the boundary plane where the scan will stop.

These vectors are shown as blue, green and orange colored arrows in the Graphical Display area of your part.



Colored arrows showing vectors

Vector	Graphical Representation
Initial Touch	Blue arrow
Direction	Green arrow
Boundary Plane	Orange arrow

Initial Touch Vector (InitVec)

The values that are displayed in the **Initial Touch Vector** row indicate the vector PC-DMIS will use to take the first touch in the scanning process.

To edit the I, J, K Initial Touch vector:

1. Double click on **InitVect** in the vector column. The **Edit Scan Item** dialog box appears.
2. Change the values.
3. Click the **OK** button. The dialog box will close.

Cut Plane Vector (CutVec)

A cut plane is used internally for DCC scanning calculations. This cut plane is derived from the Initial Touch vector, and the vector between the first and last points for the Linear Open DCC scan. Please refer to the individual scans for details on how the Cut Plane vector is derived.

End Touch Vector (EndVec)

The End Touch vector is the approach vector of the scan at the end of the row. This is used only to stop the scan or to move to the next row (in the case of a Patch scan).

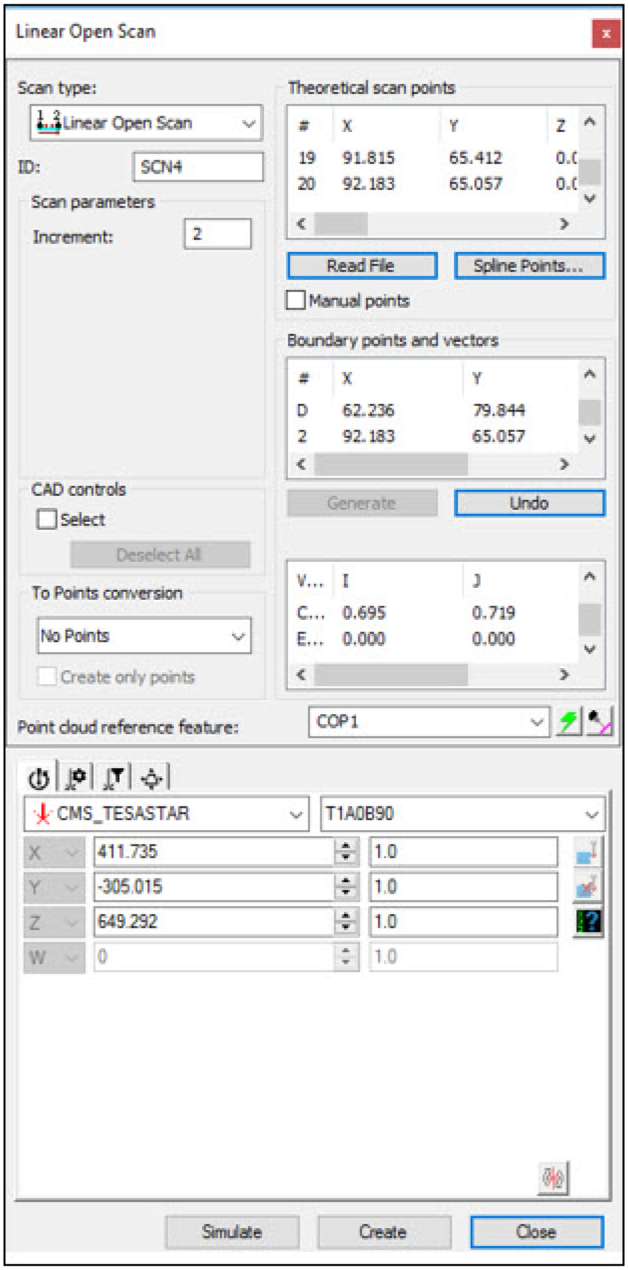
Point Cloud Reference Feature

The **Point Cloud Reference Feature** defines the Point Cloud object into which PC-DMIS places the surface data. Select the needed Pointcloud from the combo box to which data will be added. This field must be supplied, or PC-DMIS cannot create the scan.

Measure

If you select the **Measure** check and click the **Create** button, PC-DMIS will start measuring the scan immediately. If you don’t select the **Measure** check box when you click **Create**, PC-DMIS inserts a scan object into the Edit window that can be measured later. This allows you to set up a series of scans that can be inserted into the Edit window and measured at a later time.

Performing a Linear Open Advanced Scan



Linear Open Scan dialog box

The **Linear Open Scan** method scans the surface along a line. This procedure uses the starting and ending point for the line, and also includes a direction point for the calculation of the cut plane. The probe always remains within the cut plane while doing the scan.

To Create a Linear Open Scan

1. Ensure that you have a Laser probe enabled.
2. Place PC-DMIS into DCC mode.
3. Select the **Insert | Scan | Linear Open** menu item. The **Scan** dialog box appears with **Linear Open Scan** already selected in the **Scan type** list.
4. If your scan traverses multiple surfaces, consider selecting surfaces as discussed in “CAD Controls”. To access these controls, click the **Advanced >>** button in the upper-right corner of the dialog box if necessary, and then click the **Graphics** tab at the bottom.
5. If you are going to use the boundary points to help define the scan path, add the 1 point (starting point), the D point (direction to scan), and the 2 point (ending point), to the scan by following an appropriate procedure as discussed in "Boundary Points”.
6. Make any needed changes to the vectors in the **Vectors** list by double-clicking on the vector. Make any changes in the **Edit Scan Item** dialog box, and then click **OK** to return to the **Scan** dialog box.
7. Type the name of the scan in the **ID** box.
8. Select the **Measure** check box if needed.
9. Set the distance between generated theoretical points in the **Increment** box.
10. Select the method for defining the scan path from the **Read File**, **Manual Hits**, **Generate**, and **Spline Points** options.
11. If needed, you can delete individual points. To delete them, select them one at a time in the **Theoretical Path** area and press the Delete key on your keyboard.
12. If needed, make additional modifications to your scan.
13. In the **Point cloud reference feature** box, type the ID of the cloud of points object that will receive the surface data.
14. In the **Hit Type** list, you can select **Surface Point** in case you want to convert scan data into Surface Point laser commands. PC-DMIS inserts these commands into the Edit window when you click the **Create** button.

> **WARNING:** Once you mark the **Measure** check box, and click **Create**, you must keep clear of the machine. The software starts the measurement routine and the machine will move. Injury may result if you are not clear of the machine.

15. Click the **Create** button. PC-DMIS inserts the scan into the Edit window if the **Create only points** check box is not selected.

Scan Parameters

The **Increment** box in the **Scan Parameters** area lets you set the increment distance between the theoretical points when you click the **Generate** button.

Vectors

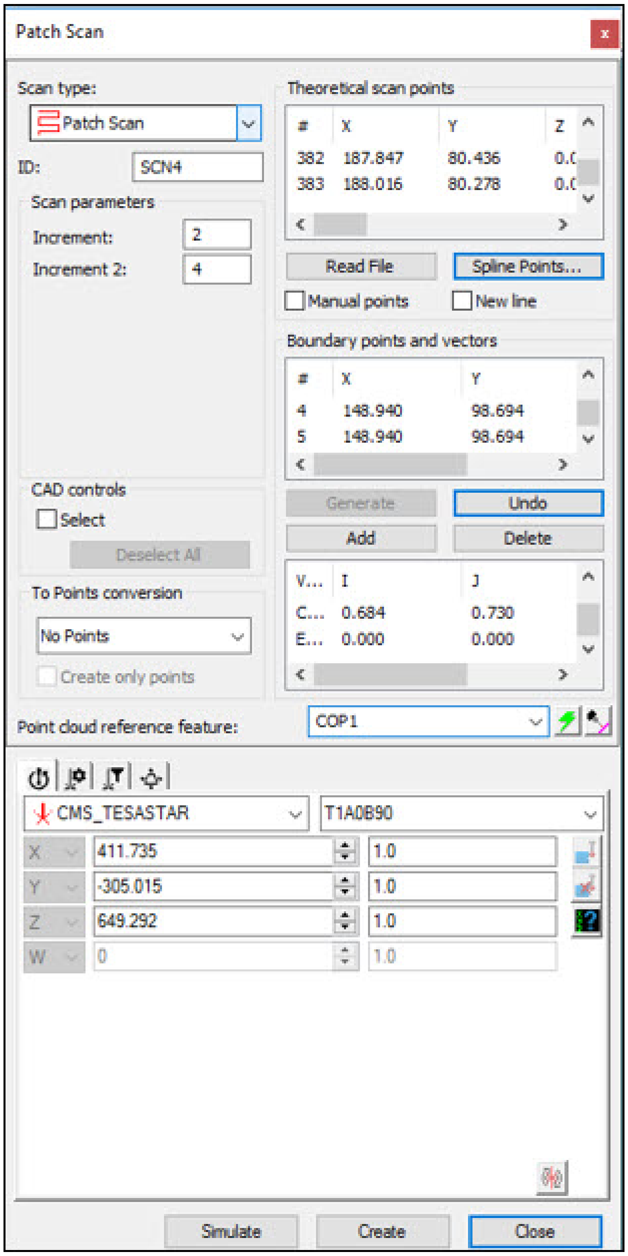
Vectors used:

- Cut Plane (CutVec)
- Initial Touch (InitVec)
- End Touch (EndVec)

See "Vectors” under "Common Functions of Scan Dialog Boxes" for additional information.

The Cut Plane vector (CutVec) is the cross product of the Initial Touch vector(InitVec) and the line between the start and end point.

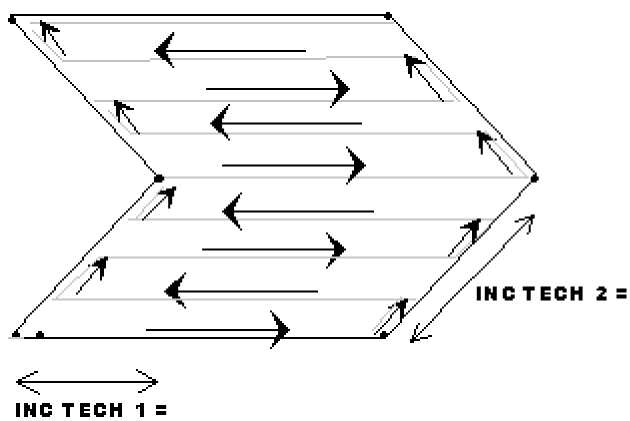
Performing a Patch Advanced Scan



Patch Scan dialog box

The Patch scan is like a series of Linear Open scans done parallel to each other.

The **Patch Scan** method scans the surface of the part based on the Scan Parameters. The probe always remains within the cut plane while doing each scan line. It uses the **Increment** value to determine the distance between points on each line. When the scan reaches the boundary at the end of a line, the scan moves to the next line by the **Increment 2** value and starts a new scan line moving in the opposite direction. The following figure describes this process.



Patch scan increment example

To Create a Patch Scan

1. Ensure that you have a Laser probe enabled.
2. Place PC-DMIS into DCC mode.
3. Select the **Insert | Scan | Patch** menu item. The **Scan** dialog box appears with **Patch Scan** already selected in the **Scan type** list.
4. Set the values for **Increment** and **Increment 2**. These determine the spacing of the points if you select the **Generate** or **Spline** buttons, or the **New Line** check box to define the scan. **Increment** defines the spacing between each point on a scan line and **Increment 2** defines the spacing between scan lines.
5. If your scan traverses multiple surfaces, consider selecting surfaces as discussed in "CAD Controls".
6. If you are going to use the boundary points to help define the scan path add the 1 point (starting point), the D point (the direction to begin scanning), the 2 point (the end point of the first line), the 3 point (to generate a minimum area), and, if desired, the 4 point (to form a square or rectangular area). This will select an area that you wish to scan. Pick these points by following an appropriate procedure as discussed in "Boundary Points".
7. Make any needed changes to the vectors in the **Vectors** list. To do this, double-click on the vector, make any changes to the **Edit Scan Item** dialog box, and then click **OK** to return to the **Scan** dialog box.
8. Type the name of the scan in the **ID** box.
9. Mark the **Measure** check box if you want to execute the scan and measure it at creation time.

10. Select the **Generate** button to generate a preview of the scan on the CAD model in the Graphic Display window. When you generate the scan, PC-DMIS starts the scan at the start point, and follows the chosen direction until it reaches the boundary point. The scan then moves back and forth scanning in rows along the chosen area, and scanning in rows at the specified increment value until it finishes the process.
11. If needed, you can delete individual points. To delete them, select them one at a time in the **Theoretical Path** area and press the Delete key on your keyboard.
12. If needed, make additional modifications to your scan.
13. In the **Point cloud reference feature** box, type the ID of the cloud of points object that will receive the surface data.
14. In the **Hit Type** list, you can select **Surface Point** in case you want to convert scan data into Surface Point laser commands. PC-DMIS inserts these commands into the Edit window when you click the **Create** button.

WARNING: Once you mark the **Measure** check box, and click **Create**, you must keep clear of the machine. The software starts the measurement routine and the machine will move. Injury may result if you are not clear of the machine.

15. Click the **Create** button. PC-DMIS inserts the scan into the Edit window if the **Create only points** check box is not selected.

Patch Scan Parameters

The **Increment** and **Increment 2** boxes described below are available when creating and measuring a **Patch** scan.

Increment

The **Increment** allows you to set the increment distance between each point when Generate or Spline/Line is used to define the scan path.

Increment 2

The **Increment 2** allows you to set the increment distance between scan lines when Generate or Spline/Line is used to define the scan path.

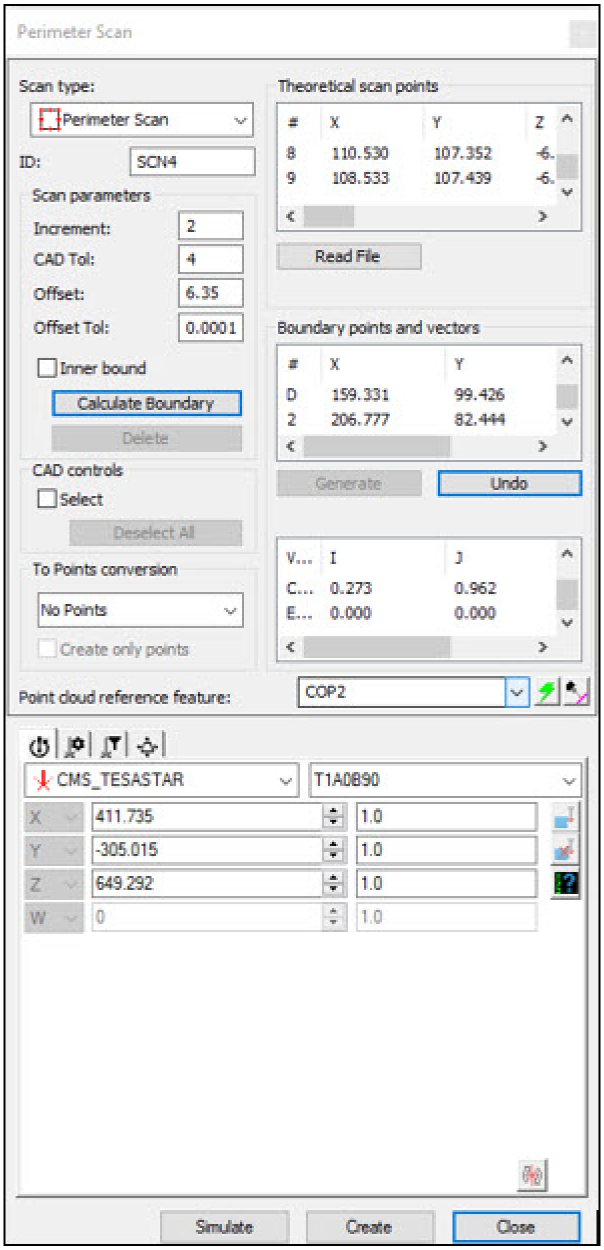
Initial Vectors

Vectors used:

- Cut Plane (CutVec)
- Initial Touch (InitVec)
- End Touch (EndVec)

The cut plane vector is derived by crossing the Initial Touch vector (InitVect) and the line between the first and second point. The cut plane vector is then set to the correct direction by using the line between the second and third points. The End Touch vector (EndVec) is the vector used to take the second boundary points and is used to jump to the second row after completing the first row.

Performing a Perimeter Advanced Scan



Perimeter Scan dialog box

The **Perimeter Scan** method scans the surface of the part based on the selected surfaces. This procedure traverses the selected surfaces within the created boundaries.

To Create a Perimeter Scan

To create a Perimeter scan:

1. Ensure that you have a Laser probe enabled.
2. Place PC-DMIS into DCC mode.
3. Select the **Insert | Scan | Perimeter** menu item. The **Scan** dialog box appears with **Perimeter Scan** already selected in the **Scan type** list.
4. Select the surface or surfaces to use to create the boundary. If you select multiple surfaces, you should select the surfaces in the same order that they are to be traversed by the scan. To select the necessary surface or surfaces:
5. Verify that the **Select** check box is selected. Each surface will be highlighted as it is selected.
6. After you select the desired surfaces, clear the **Select** check box.
7. Click on the surface near the boundary where the scan is to begin. This is the Start Point.
8. Click on the same surface a second time in the direction that the scan is to be executed. This is the Direction Point.
9. Click the point where the scan is to end. This point is *optional*. If you do not provide an End Point, the scan ends at its Start point.
10. Type the appropriate values into the **Scan parameters** area. These include the following boxes:

    - **Increment** box
    - **CAD Tol** box
    - **Offset** box
    - **Offset Tol (+/-)** box

11. Select the **Calculate Boundary** button to calculate the boundary from which the scan will be created. The red dots on the boundary indicate where the hits are taken on the perimeter scan.

NOTE: The boundary calculation should be a relatively quick process.

If the boundary does not look correct, click the **Delete** button. This will delete the boundary and allow another to be created.

If the boundary appears incorrect, it usually means that the CAD tolerance needs to be increased.

After changing the CAD tolerance, click the **Calculate Boundary** button to recalculate the boundary.

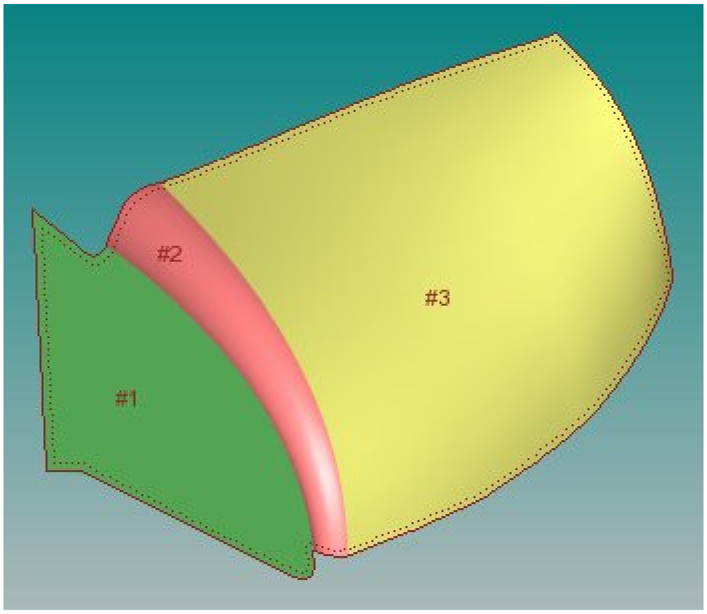
Verify that the boundary is correct before calculating a perimeter scan because it takes much longer to calculate the scan path than it does to recalculate the boundary.

12. Verify that the **Offset** value is correct.
13. Click the **Generate** button. PC-DMIS calculates the theoretical values used to execute the scan. This process involves a very time intensive algorithm. Depending on the complexity of the selected surfaces and the amount of points that are being calculated, it may take a while to compute the scan path. (A five-minute wait is common.) If the scan does not appear to be correct, you can use the **Undo** button to delete the proposed scan path. If needed, you can alter the offset tolerance and recalculate the scan.
14. If needed, you can delete individual points by selecting them one at a time in the **Theoretical Path** area and pressing the Delete key on your keyboard.
15. In the **Point cloud reference feature** box, type the ID of the cloud of points object that will receive the surface data.
16. In the **Hit Type** list, you can select **Surface Point** or **Edge Point** in case you want to convert scan data into Surface Point or Edge Point laser commands. PC-DMIS inserts these commands into the Edit window when you click the **Create** button.

WARNING: Be aware that if the **Measure** check box is marked, the machine begins to move as soon as you click **Create**. Make sure you are well clear of the machine to avoid injury.

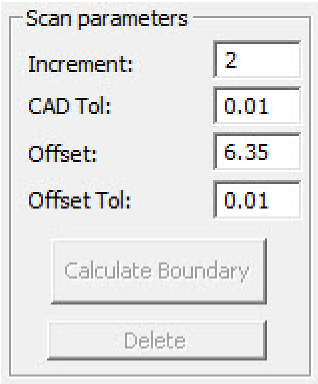
17. Click the **Create** button to store the perimeter scan in the Edit window if the **Create only points** check box is not selected. It is executed like any other scan. If you have the PC-DMIS AutoWrist method enabled but do not have any calibrated tips, PC-DMIS displays a message that informs you when it adds new probe tips that need calibration. In all other cases, PC-DMIS prompts you whether it should use the closest calibrated tip to the needed tip angle or add in a new non-calibrated tip at the needed angle.

Three surfaces have been selected. Each surface borders another, but the outside of each surface makes up the composite boundary (indicated by the solid line) The offset distance is the amount that the scan will be offset from the composite boundary (indicated by the dotted line)



Perimeter Scan Example

Perimeter Scan Parameters



Scan Parameters area

The **Scan Parameters** area of the dialog box allows various options for constructing a Perimeter Scan. These include:

Increment

The **Increment** box indicates the distance between each of the hit points on the scan.

CAD Tol

The **CAD Tol** box is useful in detecting neighboring surfaces. The larger the tolerance, the farther apart the CAD surfaces can be and still be recognized as a neighboring surface.

Offset

The **Offset** box indicates the distance in from the perimeter where the scan will be created and executed.

Offset + / -

The **Offset Tol (+/-)** box indicates the amount of allowable deviation from the offset value. It is a user supplied value.

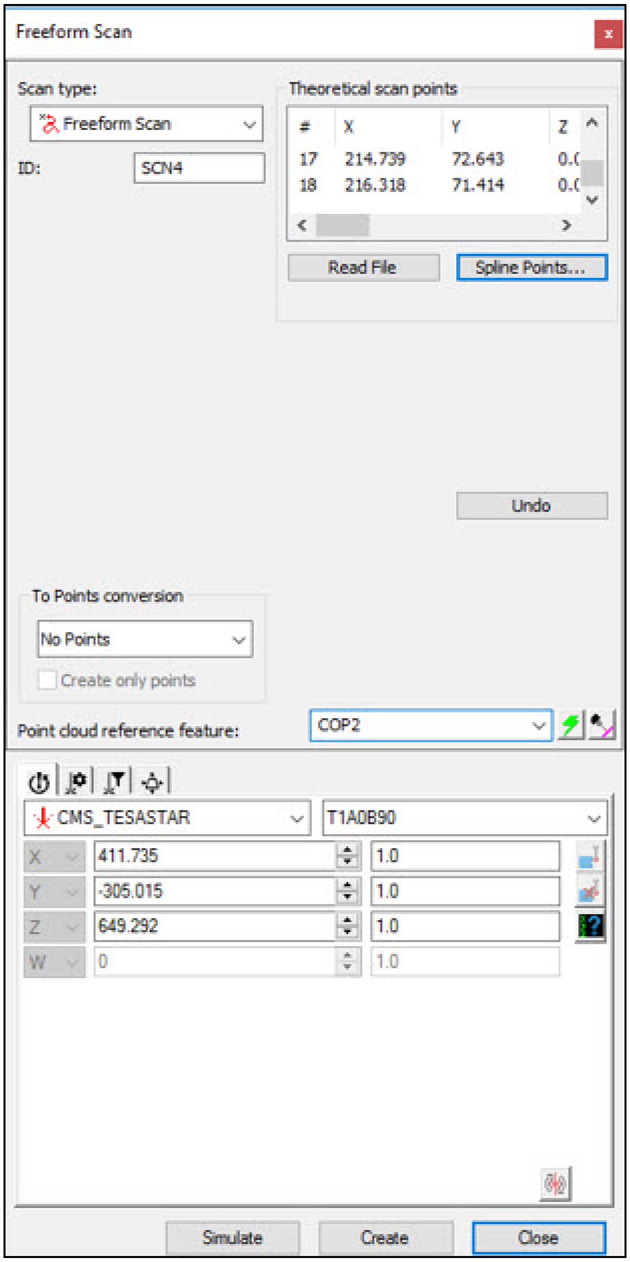
Calculate Boundary

The **Calculate Boundary** button determines the composite boundary of the input surfaces. The Calculated boundary appears as red dots in the Graphic Display window.

Delete

The **Delete** button deletes the previously created boundary.

Performing a Freeform Advanced Scan



Freeform Scan dialog box

The **Freeform Scan** method defines a scan path that is not restricted to following any particular rule set. You can define the scan path to move in any direction, including crossing back over itself.

Creating a Freeform Scan

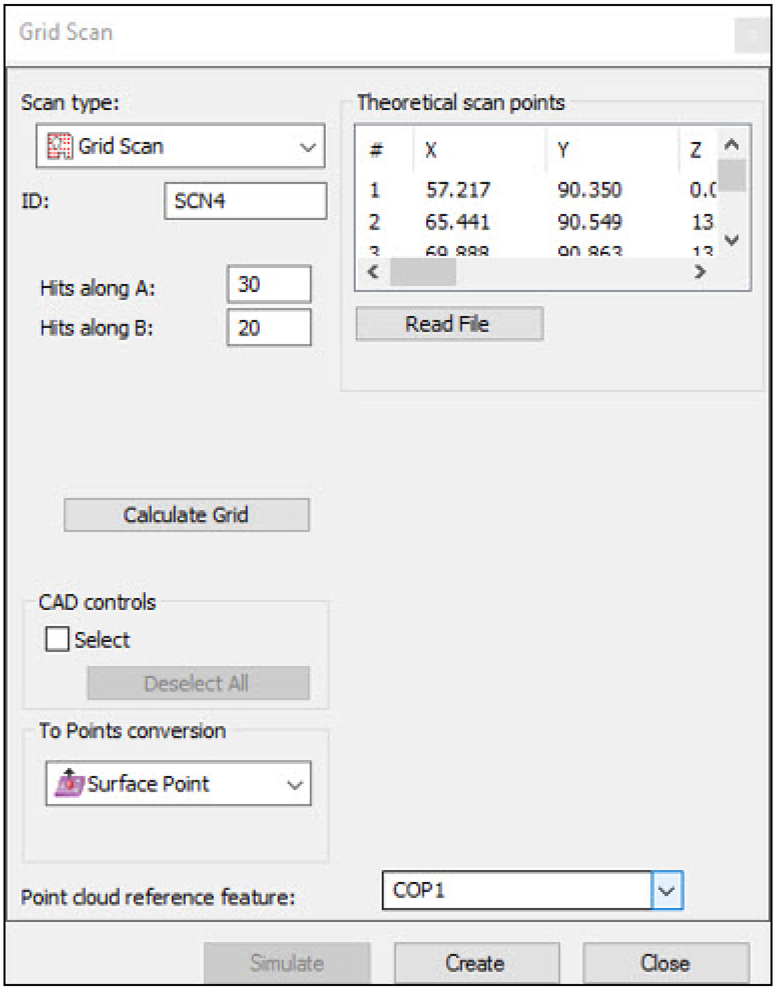
1. Place PC-DMIS into DCC mode.

2. Select the **Insert | Scan | Freeform** menu item. The **Scan** dialog box appears with the **Freeform Scan** already selected in the **Scan type** list.
3. You need to define the scan path. You can do this by using the **Read File** option or by the **Manual Points** method.
4. If needed, you can delete individual points. To delete them, select them one at a time in the **Theoretical Path** area and press the Delete key on your keyboard.
5. Once five or more **Theoretical Points**, use the **Spline Points** option to better define the path.
6. If needed, make additional modifications to your scan.
7. In the **Point cloud reference feature** box, type the ID of the cloud of points object that will receive the surface data.
8. In the **Hit Type** list, you can select **Surface Point** in case you want to convert scan data into Surface Point laser commands. PC-DMIS inserts these commands into the Edit window when you click the **Create** button.

WARNING: Once you mark the **Measure** check box, and click **Create**, you must keep clear of the machine. The software starts the measurement routine and the machine will move. Injury may result if you are not clear of the machine.

9. Click the **Create** button. PC-DMIS inserts the scan into the Edit window if the **Create only points** check box is not selected. If you have the PC-DMIS AutoWrist method enabled but don't have any calibrated tips, PC-DMIS displays a message that informs you when it adds new probe tips that need calibration. In all other cases, PC-DMIS asks you whether it should use the closest calibrated tip to the needed tip angle or add in a new non-calibrated tip at the needed angle.

Performing a Grid Advanced Scan

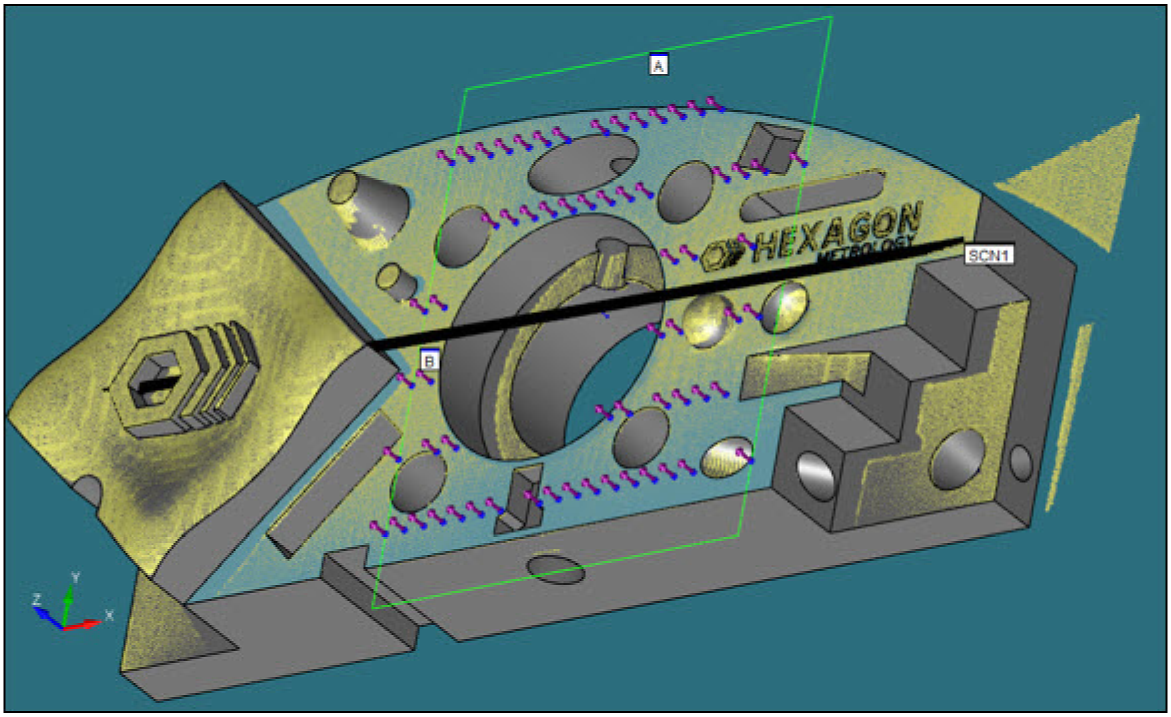


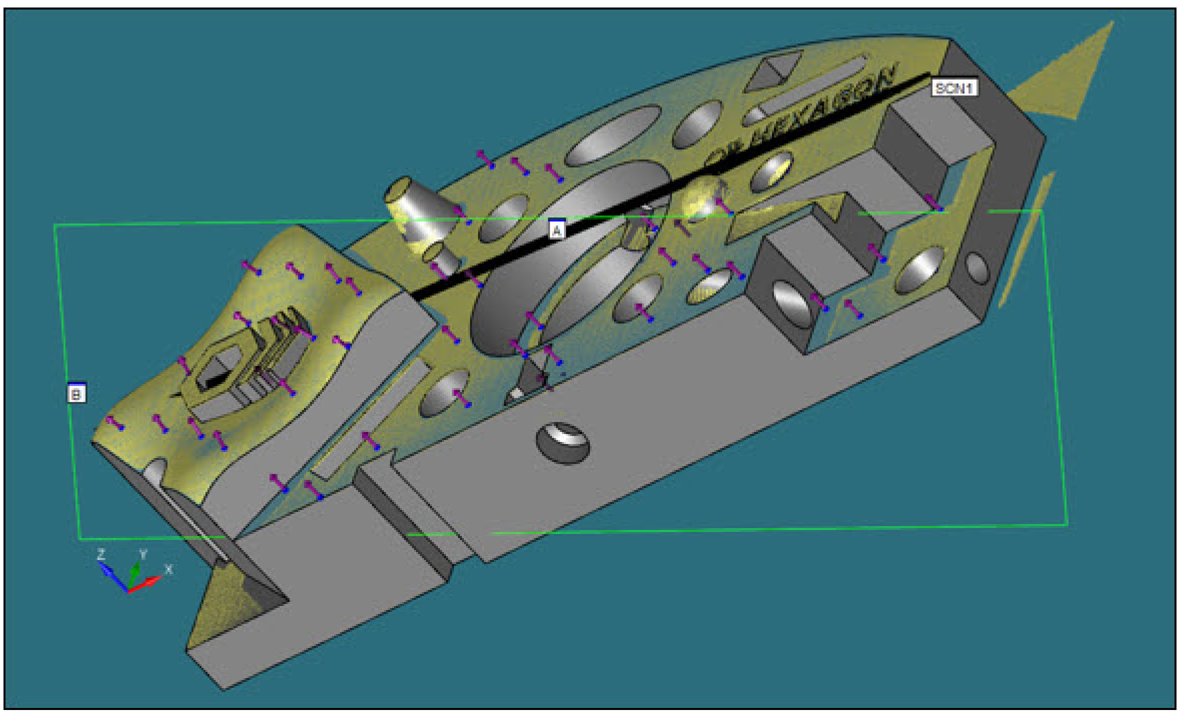
Grid Scan dialog box

The **Grid Scan** method creates a grid of points within a visible rectangle and then projects those points down on top of any selected surfaces. The rectangle and, consequently, the grid of points, depend on the orientation of the CAD model in the CAD view.

Use the **Hits along A** and **Hits along B** boxes to define how many hits inside the boundary will get spaced and dropped onto the selected surface or surfaces.

Consider the following figures that show grid Surface Points extracted from a COP:





Creating a Grid Scan

1. Ensure that you have a Laser probe enabled.

2. Place your CAD model into Solid mode.
3. Place PC-DMIS into DCC mode.
4. Select the **Insert | Scan | Grid** menu item. The **Scan** dialog box appears with the **Grid Scan** already selected in the **Scan type** list.
5. If you want to use a custom name for the grid, type the name of the grid in the **ID** box.
6. In the **Hits along A** and **Hits along B** boxes, specify how many hits in the A and B directions will get spaced and dropped onto the selected surface or surfaces.
7. Click and drag a rectangle on the screen over the surface or surfaces you want to include in your scan. This rectangle defines the boundary of the grid, which will be projected onto the CAD surface or surfaces. PC-DMIS draws points on the CAD model on any surfaces that were selected when you drew the rectangle.
8. You should select the **Select** check box in case you want to deselect some surfaces. PC-DMIS highlights the selected surfaces and draw points only on them. It does not draw points on any deselected surfaces, even if they are included in the boundary of the rectangle.
9. If you mistakenly select a surface, click on that surface a second time to deselect it. To deselect all of the highlighted surfaces at once, click the **Deselect All** button.
10. To recompute grid points (that is, to apply different A and B values on the selected surfaces), select the **Calculate Grid** button at any time.
11. In the **Point cloud reference feature** box, type the ID of the COP object from which to extract the surface data.
12. In the **Hit Type** list, **Surface Point** is the only option available since the scope of dialog box is to convert grid data into Surface Point laser commands. PC-DMIS inserts these commands into the Edit window when you click the **Create** button.
13. Click the **Create** button. PC-DMIS inserts the Surface Point laser commands into the Edit window in a collapsed `Group` command.

Performing a Manual Laser Scan on DCC Machines

Manual laser scans on DCC machines only work on FDC controllers and therefore only on Bridge machines with indexable heads. Manual laser scanning functionality is not available on horizontal arms with CW43L wrists.

To create a manual laser scan on a DCC machine:

1. Start PC-DMIS online with a laser sensor.
2. From the main menu, select **File | New** to start the machine in **Manual** mode.
3. Press the **Probe Enable** button on the jog box (you only need to press the button once, regardless of the button's state). The sensor initializes and the **Laser View**

tab appears in the Graphic Display window. The software automatically creates a COP command.

HINT: If the **Probe Toolbox** was already open, you can still change the sensor **Zoom** settings as needed.

4. Use the **Laser View** tab and position the probe over the part in range as needed.
5. On the jog box, change the **Probe Enable** option to the "Enable" state. If not, it does not collect data.
6. On the jog box, press the **Record** button to start scanning. The **Laser View** tab closes immediately, and the scanned data populates the COP object and the Graphic Display window in real time.
7. Use the jog box and move the probe over the part to scan it until you are satisfied with the data coverage.
8. To stop scanning, press the **Record** button again.
9. If necessary, press the **Probe Enable** button again to scan more data. You'll be prompted to empty the existing COP command or add new data to what is already there.
10. Repeat from Step 6 above to continue scanning.

You can also create a manual scan on a DCC machine by:

1. Follow steps 1-4 above.
2. On the jog box, change the **Probe Enable** button to the "Disabled" state.
3. On the jog box, press the **Record** button.
4. On the jog box, use the **Probe Enable** button to toggle data collection "On" and "Off".
5. On the jog box, press the **Record** button a second time to stop scanning and finalize the COP data.

Setting the Machine Speed for Scanning

To properly define your machine's speed for scanning with your laser, you need to do the following:

- VHSS must be supported by your controller. PC-DMIS uses this high speed mode by default when supported by the CMM.
- The `ScanSpeed` registry entry, found in the **Leitz** section of the PC-DMIS Settings Editor, limits the maximum scan speed value that you can send to the controller. By default, this is set to 50 mm/sec. Any value set by a SCANSPEED/

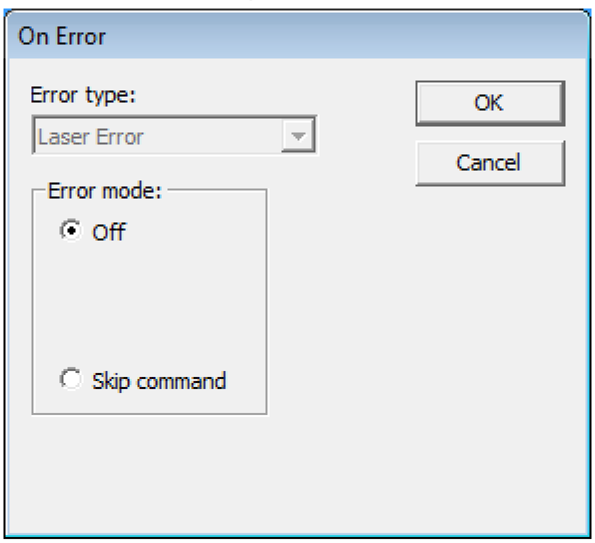
Edit window command is limited to the value of the ScanSpeed registry entry. This value can be increased accordingly the CMM limits.

- By default, PC-DMIS's **Acceleration** value, located in the **Opt. Probe** tab of the **Parameter Settings** dialog box, is set very low (10 mm/sec). To get higher scan speeds, you should increase this value to a desired value up to the limits allowed by your machine. To access this tab, select the **Edit | Preferences | Parameters** menu item and then click on the **| Opt. Probe** tab.

Handling Laser Sensor Errors with ONERROR

You can tell PC-DMIS to skip commands that generate certain laser sensor-related errors during execution by using the ONERROR command. The command only applies to the default Asynchronous Execution mode. To show the **On Error** dialog box, choose **Insert | Flow Control Command | On Error**. Choose **Skip command** and then click **OK** to insert the ONERROR command.

On Error Dialog Box



On Error dialog box

The information in this topic is specific to Laser configurations. For more information on this dialog box and how it applies to tactile probes, see the "Branching On an Error" topic of the PC-DMIS Core documentation.

The **Error mode** area contains two options:

- **Off** - The command is not skipped. If PC-DMIS encounters an error in this mode, execution stops completely.

- **Skip** - Execution continues and PC-DMIS skips commands if they generate any of the following errors:
    - No laser stripes found for feature execution
    - No scan data
    - Feature calculation error

If PC-DMIS encounters any other laser errors, it stops execution and ignores the `ONERROR` command.

The command has the following syntax in the **Edit** window's Command mode:

```
ONERROR/LASER_ERROR,TOG1
```

TOG1 = This switches between SKIP or OFF.

Index

Glossary

C

CCD: Charge Coupled Device - This is one of the two main types of image sensors used in digital cameras.

Cloud of Points: The Cloud of Points command is a container for XYZ coordinate data. The data can be input from an external file, or it can come directly from a laser sensor through the referring scan command(s).

COP: The Cloud of Points command is a container for XYZ coordinate data. The data can be input from an external file, or it can come directly from a laser sensor through the referring scan command(s).

E

Exposure: This parameter controls the exposure of the Laser sensor.

G

Gauge side point: In a Flush and Gap Auto Feature, this is the point on the gauge surface side indicating where the flush should be measured. (also called the gauge point)

L

LWM: Laser Wrist Map

M

Master side point: In a Flush and Gap Auto Feature, this is the point on the master side surface where the flush is to be measured.

Mesh: A mesh is a set of vertices and triangles which are combined using a best fit algorithm to represent a 3D part shape.

milli-pixel: 1 milli-pixel = 0.001 pixel

O

Overscan: This parameter controls how far beyond the nominal feature's dimensions the probe will scan along both the major and minor axis of the feature.

P

Pointcloud: A Pointcloud is a collection of data points used to define a feature on a CAD model.

R

Row Overlap: This parameter controls how far each pass will overlap with the previous pass.

S

Sensor Frequency: This parameter controls the internal sensor frequency of the probe. The value that is displayed is sensor pulses per second.

Surface CAD Model: A surface CAD model only has surfaces, and it does not create a solid. Some examples of this would be, a plane feature, or a cylinder surface where there is no closed volume.